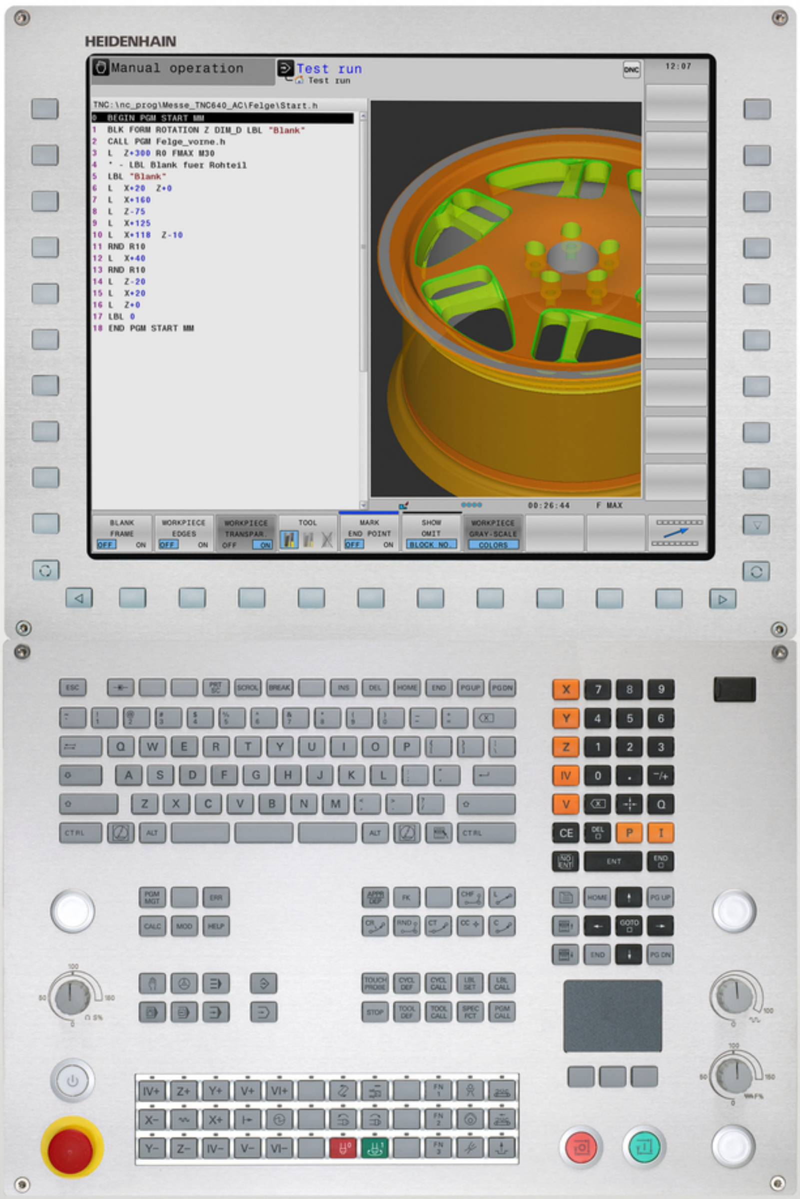


HEIDENHAIN

TNC 640

User's Manual
Cycle Programming

NC Software
340590-05
340591-05
340595-05

English (en)
1/2015

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.

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

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 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:

All TNC functions that have no connection with cycles are described in the User's Manual of the TNC 640. Please contact HEIDENHAIN if you require a copy of this User's Manual.

ID of User's Manual for conversational programming: 892904-xx.

ID of User’s Manual for DIN/ISO programming: 892910-xx.

Software options

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

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

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

Touch Probe Functions (option 17)	
Touch probe functions	**Touch probe cycles:** ■ Compensation of tool misalignment in automatic mode ■ Datum setting in the **Manual Operation** mode ■ Datum setting in automatic mode ■ Automatically measuring workpieces ■ Tools can be measured automatically

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

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

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

DXF Converter (Option #42)	
DXF converter	■ 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	■ 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	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics

Mill-Turning (Option #50)	
Milling and turning modes	**Functions:** ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning cycles

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	■ 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	■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of TCP (**T**ool **C**enter **P**oint)

Position Adaptive Control – PAC (Option #142)	
Adaptive position control	■ Changing of the control parameters depending on the position of the axes in the working space ■ Changing of the control parameters depending on the speed or acceleration of an axis

Load Adaptive Control – LAC (Option #143)	
Adaptive load control	■ Automatic determination of workpiece weight and frictional forces ■ Changing of control parameters depending on the actual mass of the workpiece

Active Chatter Control – ACC (Option #145)	
Active chatter control	Fully automatic function for chatter control during machining

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the **F**eature **C**ontent **L**evel 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** softkey

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, some of which have not been available in previous software versions. Within a cycle, they are always provided at the end of the cycle definition. You will find an overview of the optional Q parameters that have been added with this software version in the "New and changed cycle functions of software 34059x-05" section. You can choose whether to define optional parameters or delete them with the NO ENT key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing programs after a software update, you can include optional Q parameters in cycles when needed. The following steps describe how this is done:

To insert optional Q parameters in existing programs:

- Call the cycle definition
- Press the right arrow key until the new Q parameters are displayed
- Apply the default value or enter a value
- To transfer the new Q parameter, exit the menu by pressing the right arrow key once again or by pressing END
- If you do not wish to apply the new Q parameter, press the NO ENT key

Compatibility

The majority of part programs created on older HEIDENHAIN contouring controls (TNC 150 B and higher) can be executed with this new software version of the TNC 640. Even if new, optional parameters ("Optional parameters") have been added to existing cycles, you can normally continue running your programs as usual. This is achieved by using the stored default value. The other way round, if a program created with a new software version is to be run on an older control, you can delete the respective optional Q parameters from the cycle definition with the NO ENT key. In this way you can ensure that the program will be downward compatible. If NC blocks contain invalid elements, the TNC will mark them as ERROR blocks when the file is opened.

New cycle functions of software 34059x-04

- The character set of the fixed cycle 225 Engraving was expanded by more characters and the diameter sign see "ENGRAVING (Cycle 225, DIN/ISO: G225)", page 300
- New fixed cycle 275 Trochoidal Milling see "TROCHOIDAL SLOT (Cycle 275, DIN ISO G275)", page 211
- New fixed cycle 233 Face Milling see "FACE MILLING (Cycle 233, DIN/ISO: G233)", page 167
- In Cycle 205 Universal Pecking you can now use parameter Q208 to define a feed rate for retraction see "Cycle parameters", page 92
- In the thread milling cycles 26x an approaching feed rate was introduced see "Cycle parameters", page 119
- The parameter Q305 NUMBER IN TABLE was added to Cycle 404 see "Cycle parameters", page 462
- In the drilling cycles 200, 203 and 205 the parameter Q395 DEPTH REFERENCE was introduced in order to evaluate the T ANGLE see "Cycle parameters", page 92
- Cycle 241 SINGLE-LIP DEEP HOLE DRILLING was expanded by several input parameters see "SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)", page 97
- The probing cycle 4 MEASURING IN 3-D was introduced see "MEASURING IN 3-D (Cycle 4)", page 567

New and changed cycle functions of software 34059x-05

- New Cycle 880 GEAR HOBBING (software option 50), see "GEAR HOBBING (Cycle 880, DIN/ISO: G880)", page 425
- New Cycle 292 CONTOUR TURNING INTERPOLATION (software option 96), see "CONTOUR TURNING INTERPOLATION (Cycle 292, DIN/ISO: G292, software option 96)", page 286
- New Cycle 291 COUPLING TURNING INTERPOLATION (software option 96), see "COUPLING TURNING INTERPOLATION (Cycle 291, DIN/ISO: G291, software option 96)", page 295
- New Load Adaptive Control (LAC) cycle for the load-dependent adaptation of control parameters (software option 143), see "ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143)", page 309
- Cycle 270: CONTOUR TRAIN DATA was added to the cycle package (software option 19), see "CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)", page 210
- Cycle 39 CYLINDER SURFACE (software option 1) Contour was added to the cycle package, see "CYLINDER SURFACE (Cycle 39, DIN/ISO: G139, software option 1)", page 232
- The character set of the fixed cycle 225 Engraving was expanded by the CE, ß and @ characters and the system time, see "ENGRAVING (Cycle 225, DIN/ISO: G225)", page 300
- Cycles 252 to 254 were expanded by the optional parameter Q439, see "Cycle parameters", page 148
- Cycle 22 was expanded by the optional parameters Q401 and Q404, see "ROUGHING (Cycle 22, DIN/ISO: G122)", page 199
- Cycles 841, 842, 851 and 852 were expanded by the plunging feed rate Q488, see "Cycle parameters", page 372
- Cycle 484 was expanded by the optional parameter Q536, see "Calibrate the wireless TT 449 (Cycle 484, DIN/ISO: G484 Touch Probe Functions)", page 619
- Eccentric turning with Cycle 800 is possible with option 50, see "ADAPT ROTARY COORDINATE SYSTEM(Cycle 800, DIN/ISO: G800)", page 322

Contents

1

Fundamentals / Overviews

1.1 Introduction

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and several special functions are also available as cycles. Most cycles use Q parameters as transfer parameters.

> **Danger of collision!**
>
> Cycles sometimes execute extensive operations. For safety reasons, you should run a graphical program test before machining.

> If you use indirect parameter assignments in cycles with numbers greater than 200 (e.g. **Q210 = Q1**), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. **Q210**) directly in such cases.
>
> If you define a feed-rate parameter for fixed cycles greater than 200, then instead of entering a numerical value you can use soft keys to assign the feed rate defined in the **TOOL CALL** block (**FAUTO** soft key). You can also use the feed-rate alternatives **FMAX** (rapid traverse), **FZ** (feed per tooth) and **FU** (feed per rev), depending on the respective cycle and the function of the feed-rate parameter.
>
> Note that, after a cycle definition, a change of the **FAUTO** feed rate has no effect, because internally the TNC assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.
>
> If you want to delete a block that is part of a cycle, the TNC asks you whether you want to delete the whole cycle.

1.2 Available Cycle Groups

Overview of fixed cycles

CYCL DEF

- The soft-key row shows the available groups of cycles

Cycle group	Soft key	Page
Cycles for pecking, reaming, boring and counterboring	DRILLING/ THREAD	74
Cycles for tapping, thread cutting and thread milling	DRILLING/ THREAD	104
Cycles for milling pockets, studs and slots and for face milling	POCKETS/ STUDS/ SLOTS	140
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.	254
Subcontour List (SL) cycles, which allow the machining of contours consisting of several overlapping subcontours, as well as cycles for cylinder surface machining and for trochoidal milling	SL CYCLES	222
Cycles for producing point patterns, such as circular or linear hole patterns	PATTERN	178
Cycles for turning and gear hobbing	TURNING	316
Special cycles such as dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning , ascertaining the load	SPECIAL CYCLES	278

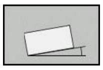
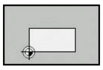
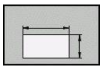
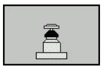
▷

- If required, switch to machine-specific fixed cycles. These fixed cycles can be integrated by your machine tool builder.

Overview of touch probe cycles

TOUCH PROBE

- The soft-key row shows the available groups of cycles

Cycle group	Soft key	Page
Cycles for automatic measurement and compensation of workpiece misalignment		448
Cycles for automatic workpiece presetting		470
Cycles for automatic workpiece inspection		522
Special cycles	SPECIAL CYCLES	564
Touch probe calibration	CALIBRATE TS	571
Cycles for automatic kinematics measurement	KINEMATICS	448
Cycles for automatic tool measurement (enabled by the machine tool builder)		612

▷

- If required, switch to machine-specific touch probe cycles. These touch probe cycles can be integrated by your machine tool builder.

2

Using Fixed Cycles

2.1 Working with fixed cycles

Machine-specific cycles

In addition to the HEIDENHAIN cycles, many machine tool builders offer their own cycles in the TNC. These cycles are available in a separate cycle-number range:

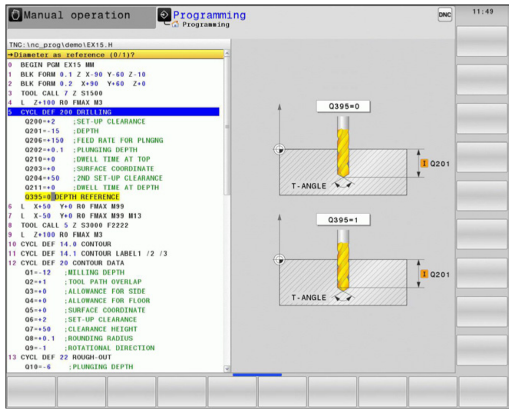
- Cycles 300 to 399
  Machine-specific cycles that are to be defined through the **CYCLE DEF** key
- Cycles 500 to 599
  Machine-specific touch probe cycles that are to be defined through the **TOUCH PROBE** key

> Refer to your machine manual for a description of the specific function.

Sometimes machine-specific cycles use transfer parameters that HEIDENHAIN already uses in standard cycles. The TNC executes DEF-active cycles as soon as they are defined (see "Calling a cycle", page 56). It executes CALL-active cycles only after they have been called (see "Calling a cycle", page 56). When DEF-active cycles and CALL-active cycles are used simultaneously, it is important to prevent overwriting of transfer parameters already in use. Use the following procedure:

- ▶ As a rule, always program DEF-active cycles before CALL-active cycles
- ▶ If you do want to program a DEF-active cycle between the definition and call of a CALL-active cycle, do it only if there is no common use of specific transfer parameters

Defining a cycle using soft keys



CYCL DEF

- The soft-key row shows the available groups of cycles

DRILLING/ THREAD

- Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles

262

- Select the cycle, e.g. THREAD MILLING. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the **ENT** key
- The TNC ends the dialog when all required data has been entered

Defining a cycle using the GOTO function

CYCL DEF

- The soft-key row shows the available groups of cycles

GOTO

- The TNC opens the smartSelect selection window with an overview of the cycles
- Choose the desired cycle with the arrow keys or mouse. The TNC then initiates the cycle dialog as described above

Example NC blocks

7 CYCL DEF 200 DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=3	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q211=0.25	;DWELL TIME AT BOTTOM

Calling a cycle

Prerequisites

The following data must always be programmed before a cycle call:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF)

For some cycles, additional prerequisites must be observed. They are detailed in the descriptions for each cycle.

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:

- Cycle 220 for circular hole patterns and Cycle 221 for linear hole patterns
- SL Cycle 14 CONTOUR GEOMETRY
- SL Cycle 20 CONTOUR DATA
- Cycle 32 TOLERANCE
- Coordinate transformation cycles
- Cycle 9 DWELL TIME
- Cycle 239 Load Adaptive Control (LAC)
- Touch probe cycles

You can call all other cycles with the functions described as follows.

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that was programmed last before the CYCL CALL block.

CYCL CALL

- ▶ To program the cycle call, press the **CYCL CALL** key
- ▶ Press the **CYCL CALL M** soft key to enter a cycle call
- ▶ If necessary, enter the miscellaneous function M (for example **M3** to switch the spindle on), or end the dialog by pressing the **END** key

Calling a cycle with CYCL CALL PAT

The **CYCL CALL PAT** function calls the most recently defined fixed cycle at all positions that you defined in a PATTERN DEF pattern definition (see "PATTERN DEF pattern definition", page 62) or in a point table (see "Point tables", page 69).

Calling a cycle with CYCL CALL POS

The **CYCL CALL POS** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that you defined in the **CYCL CALL POS** block.

Using positioning logic the TNC moves to the position defined in the **CYCL CALL POS** block.

- If the tool's current position in the tool axis is greater than the top surface of the workpiece (Q203), the TNC moves the tool to the programmed position first in the machining plane and then in the tool axis.
- If the tool's current position in the tool axis is below the top surface of the workpiece (Q203), the TNC moves the tool to the programmed position first in the tool axis to the clearance height and then in the working plane to the programmed position.

> Three coordinate axes must always be programmed in the **CYCL CALL POS** block. With the coordinate in the tool axis you can easily change the starting position. It serves as an additional datum shift.
>
> The feed rate most recently defined in the **CYCL CALL POS** block applies only to traverse to the start position programmed in this block.
>
> As a rule, the TNC moves without radius compensation (R0) to the position defined in the **CYCL CALL POS** block.
>
> If you use **CYCL CALL POS** to call a cycle in which a start position is defined (for example Cycle 212), then the position defined in the cycle serves as an additional shift of the position defined in the **CYCL CALL POS** block. You should therefore always define the start position to be set in the cycle as 0.

Calling a cycle with M99/89

The **M99** function, which is active only in the block in which it is programmed, calls the last defined fixed cycle once. You can program **M99** at the end of a positioning block. The TNC moves to this position and then calls the last defined fixed cycle.

If the TNC is to run the cycle automatically after every positioning block, program the first cycle call with **M89**.

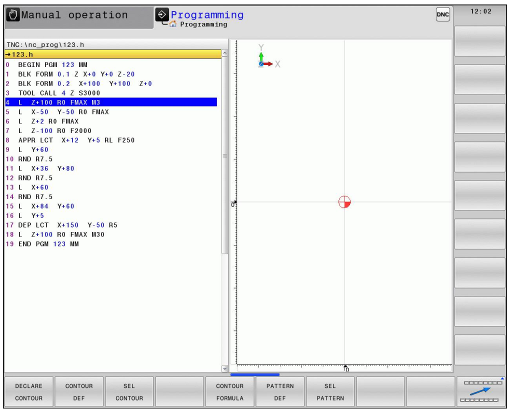
To cancel the effect of **M89**, program:

- **M99** in the positioning block in which you move to the last starting point, or
- Use **CYCL DEF** to define a new fixed cycle

2.2 Program defaults for cycles

Overview

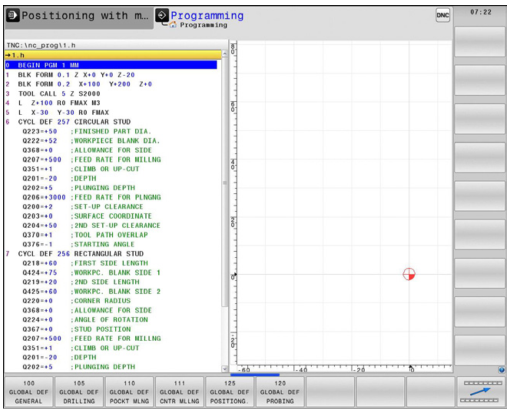
All Cycles 20 to 25, as well as all of those with numbers 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200,** which you must enter for each cycle definition. The **GLOBAL DEF** function gives you the possibility of defining these cycle parameters once at the beginning of the program, so that they are effective globally for all fixed cycles used in the program. In the respective fixed cycle you then simply link to the value defined at the beginning of the program.



The following GLOBAL DEF functions are available:

Machining patterns	Soft key	Page
GLOBAL DEF COMMON Definition of generally valid cycle parameters	100 GLOBAL DEF GENERAL	60
GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	105 GLOBAL DEF DRILLING	60
GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	110 GLOBAL DEF POCKT MLNG	60
GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	111 GLOBAL DEF CNTR MLLNG	61
GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	125 GLOBAL DEF POSITIONG.	61
GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	120 GLOBAL DEF PROBING	61

Entering GLOBAL DEF



- Select the Programming and Editing operating mode

SPEC FCT
- Press the special functions key

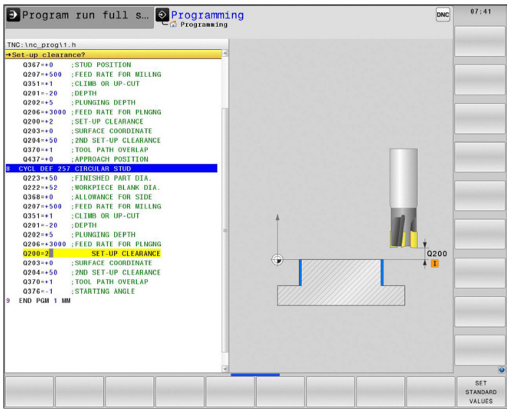
PROGRAM DEFAULTS
- Select the functions for program defaults

GLOBAL DEF
- Select **GLOBAL DEF** functions

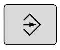
100 GLOBAL DEF GENERAL
- Select the desired GLOBAL DEF function, e.g. **GLOBAL DEF COMMON**
- Enter the required definitions, and confirm each entry with the ENT key

Using GLOBAL DEF information

If you have entered the corresponding GLOBAL DEF functions at the beginning of the program, then you can link to these globally valid values when defining any fixed cycle.



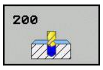
Proceed as follows:



- Select the Programming and Editing operating mode

- Select fixed cycles

- Select the desired group of cycles, for example: drilling cycles



- Select the desired cycle, e.g. **DRILLING**
- The TNC displays the **SET STANDARD VALUES** soft key, if there is a global parameter for it

- Press the **SET STANDARD VALUES** soft key. The TNC enters the word **PREDEF** (predefined) in the cycle definition. You have now created a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program

!

Danger of collision!

Please note that later changes to the program settings affect the entire machining program, and can therefore change the machining procedure significantly.

If you enter a fixed value in a fixed cycle, then this value will not be changed by the **GLOBAL DEF** functions.

Global data valid everywhere

- **Set-up clearance**: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis
- **2nd set-up clearance:** Position to which the TNC positions the tool at the end of a machining step. The next machining position is approached at this height in the machining plane
- **F positioning:** Feed rate at which the TNC traverses the tool within a cycle
- **F retraction**: Feed rate at which the TNC retracts the tool

> The parameters are valid for all fixed cycles with numbers greater than 2xx.

Global data for drilling operations

- **Retraction rate for chip breaking:** Value by which the TNC retracts the tool during chip breaking
- **Dwell time at depth**: Time in seconds that the tool remains at the hole bottom
- **Dwell time at top**: Time in seconds that the tool remains at the set-up clearance

> The parameters apply to the drilling, tapping and thread milling cycles 200 to 209, 240, and 262 to 267.

Global data for milling operations with pocket cycles 25x

- **Overlap factor**: The tool radius multiplied by the overlap factor equals the lateral stepover
- **Climb or up-cut**: Select the type of milling
- **Plunging type**: Plunge into the material helically, in a reciprocating motion, or vertically

> The parameters apply to milling cycles 251 to 257.

Global data for milling operations with contour cycles

- **Set-up clearance**: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis
- **Clearance height**: Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle)
- **Overlap factor**: The tool radius multiplied by the overlap factor equals the lateral stepover
- **Climb or up-cut**: Select the type of milling

> The parameters apply to SL cycles 20, 22, 23, 24 and 25.

Global data for positioning behavior

- **Positioning behavior**: Retraction in the tool axis at the end of the machining step: Return to the 2nd set-up clearance or to the position at the beginning of the unit

> The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Global data for probing functions

- **Set-up clearance**: Distance between stylus and workpiece surface for automated approach of the probing position
- **Clearance height:** The coordinate in the touch probe axis to which the TNC traverses the touch probe between measuring points, if the **Move to clearance height** option is activated
- **Move to clearance height**: Select whether the TNC moves the touch probe to the set-up clearance or clearance height between the measuring points

> Applies to all Touch Probe Cycles 4xx.

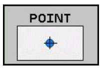
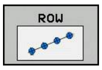
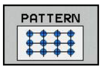
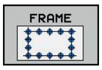
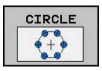
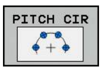
2.3 PATTERN DEF pattern definition

Application

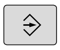
You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. As with the cycle definitions, support graphics that illustrate the respective input parameter are also available for pattern definitions.

> **!** **PATTERN DEF** is to be used only in connection with the tool axis Z.

The following machining patterns are available:

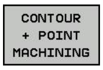
Machining patterns	Soft key	Page
POINT Definition of up to any 9 machining positions		64
ROW Definition of a single row, straight or rotated		64
PATTERN Definition of a single pattern, straight, rotated or distorted		65
FRAME Definition of a single frame, straight, rotated or distorted		66
CIRCLE Definition of a full circle		67
PITCH CIRCLE Definition of a pitch circle		68

Entering PATTERN DEF

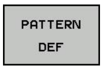


- Select the **Programming** mode of operation

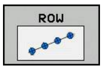
- Press the special functions key



- Select the functions for contour and point machining



- Open a **PATTERN DEF** block



- Select the desired machining pattern, e.g. a single row
- Enter the required definitions, and confirm each entry with the ENT key

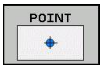
Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function "Calling a cycle", page 56. The TNC then performs the most recently defined machining cycle on the machining pattern you defined.

> A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
>
> You can use the mid-program startup function to select any point at which you want to start or continue machining (see User's Manual, Test Run and Program Run sections)see "Any entry into program (mid-program startup)".

Defining individual machining positions

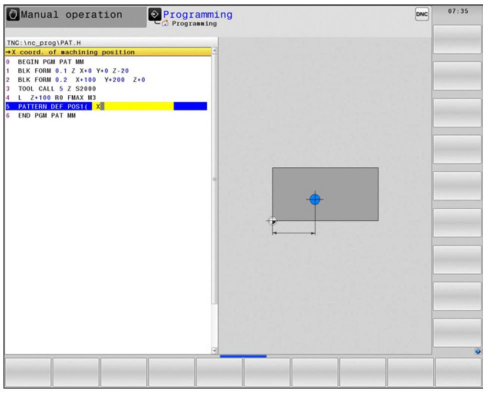
> You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.
>
> If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **X coord. of machining position** (absolute): Enter X coordinate
- **Y coord. of machining position** (absolute): Enter Y coordinate
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

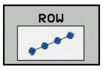
NC blocks

10 L Z+100 R0 FMAX
11 PATTERN DEF POS1 (X+25 Y+33.5 Z+0) POS2 (X+50 Y +75 Z+0)



Defining a single row

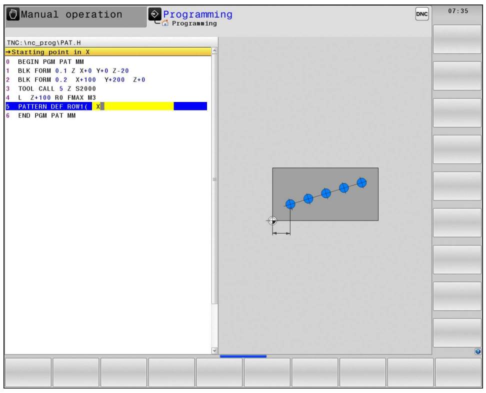
> If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Starting point in X** (absolute): Coordinate of the starting point of the row in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the row in the Y axis
- **Spacing of machining positions (incremental)**: Distance between the machining positions. You can enter a positive or negative value
- **Number of repetitions**: Total number of machining operations
- **Rot. position of entire pattern (absolute)**: Angle of rotation around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

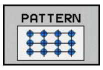
NC blocks

10 L Z+100 R0 FMAX
11 PATTERN DEF ROW1 (X+25 Y+33.5 D+8 NUM5 ROT+0 Z +0)



Defining a single pattern

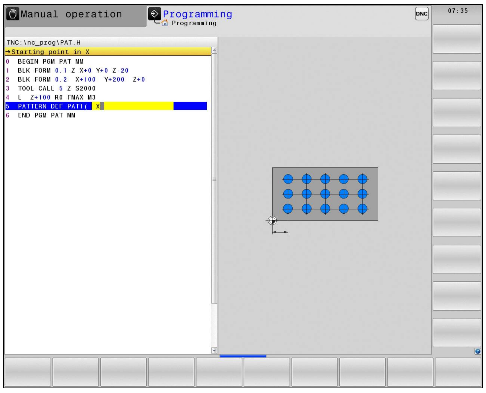
> If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
>
> The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotated position** of the entire pattern.



- **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- **Spacing of machining positions X (incremental)**: Distance between the machining positions in the X direction. You can enter a positive or negative value
- **Spacing of machining positions Y (incremental)**: Distance between the machining positions in the Y direction. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of lines**: Total number of rows in the pattern
- **Rot. position of entire pattern (absolute)**: Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Rotary pos. minor ax.**: Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Workpiece surface coordinate** (absolute): Enter Z coordinate at which machining is to begin

NC blocks

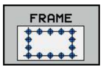
```
10 L Z+100 R0 FMAX
11 PATTERN DEF PAT1 (X+25 Y+33.5
   DX+8 DY+10 NUMX5 NUMY4 ROT+0
   ROTX+0 ROTY+0 Z+0)
```



Defining individual frames

If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

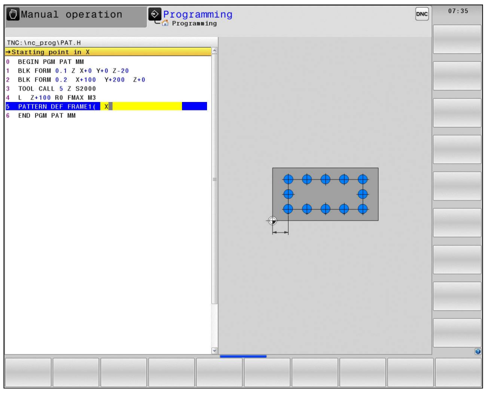
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotated position** of the entire pattern.



- **Starting point in X** (absolute): Coordinate of the starting point of the frame in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the frame in the Y axis
- **Spacing of machining positions X (incremental)**: Distance between the machining positions in the X direction. You can enter a positive or negative value
- **Spacing of machining positions Y (incremental)**: Distance between the machining positions in the Y direction. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of lines**: Total number of rows in the pattern
- **Rot. position of entire pattern (absolute)**: Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value
- **Rotary pos. minor ax.**: Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

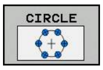
NC blocks

```
10 L Z+100 R0 FMAX
11 PATTERN DEF FRAME1
   (X+25 Y+33.5 DX+8 DY+10 NUMX5
   NUMY4 ROT+0 ROTX+0 ROTY+0 Z
   +0)
```



Defining a full circle

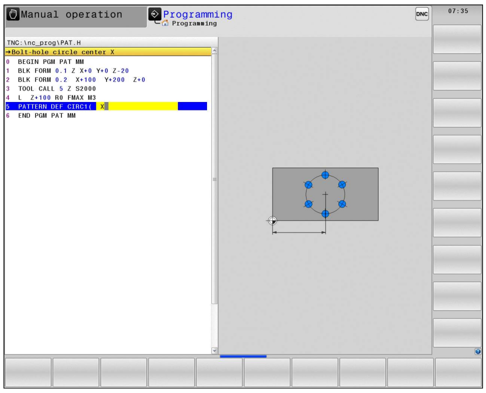
> If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Number of repetitions**: Total number of machining positions on the circle
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

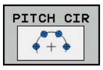
NC blocks

```
10 L Z+100 R0 FMAX
11 PATTERN DEF CIRC1
   (X+25 Y+33 D80 START+45 NUM8 Z
   +0)
```



Defining a pitch circle

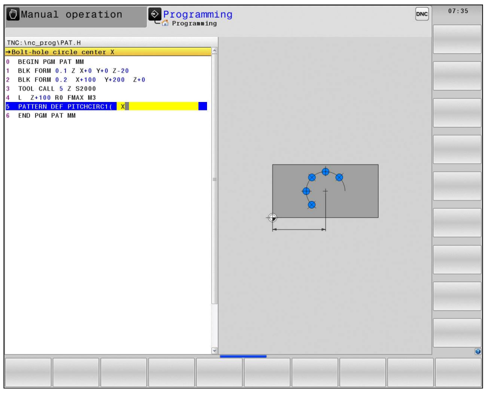
> If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- **Starting angle**: Polar angle of the first machining position. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Stepping angle/end angle**: Incremental polar angle between two machining positions. You can enter a positive or negative value As an alternative you can enter the end angle (switch via soft key).
- **Number of repetitions**: Total number of machining positions on the circle
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

NC blocks

```
10 L Z+100 R0 FMAX
11 PATTERN DEF PITCHCIRC1
   (X+25 Y+33 D80 START+45 STEP30
   NUM8 Z+0)
```



2.4 Point tables

Application

You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table

- Select the **Programming** mode of operation

PGM MGT

- Call the file manager: Press the **PGM MGT** key.

FILE NAME?

ENT

- Enter the name and file type of the point table and confirm your entry with the **ENT** key.

MM

- Select the unit of measure: Press the **MM** or **INCH** soft key. The TNC changes to the program blocks window and displays an empty point table.

INSERT LINE

- With the **INSERT LINE** soft key, insert new lines and enter the coordinates of the desired machining position.

Repeat the process until all desired coordinates have been entered.

> The name of the point table must begin with a letter.
>
> Use the soft keys **X OFF/ON**, **Y OFF/ON**, **Z OFF/ON** (second soft-key row) to specify which coordinates you want to enter in the point table.

Hiding single points from the machining process

In the **FADE** column of the point table you can specify if the defined point is to be hidden during the machining process.

- In the table, select the point to be hidden
- Select the **FADE** column

ENT
- Activate hiding, or

NO ENT
- Deactivate hiding

Selecting a point table in the program

In the **Programming** mode of operation, select the program for which you want to activate the point table:

PGM CALL
- Press the **PGM CALL** key to call the function for selecting the point table

POINT TABLE
- Press the **POINT TABLE** soft key

Enter the name of the point table and confirm your entry with the **END** key. If the point table is not stored in the same directory as the NC program, you must enter the complete path.

Example NC block

```
7 SEL PATTERN "TNC:\DIRKT5\MUST35.PNT"
```

Calling a cycle in connection with point tables

> With **CYCL CALL PAT** the TNC runs the point table that you last defined (even if you defined the point table in a program that was nested with **CALL PGM).**

If you want the TNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with **CYCLE CALL PAT**:

CYCL CALL

- To program the cycle call, press the **CYCL CALL** key
- Press the **CYCL CALL PAT** soft key to call a point table
- Enter the feed rate at which the TNC is to move from point to point (if you make no entry the TNC will move at the last programmed feed rate; **FMAX** is not valid)
- If required, enter a miscellaneous function M, then confirm with the **END** key

The TNC retracts the tool to the safety clearance between the starting points. Depending on which is greater, the TNC uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103.

Effect of the point tables with SL cycles and Cycle 12

The TNC interprets the points as an additional datum shift.

Effect of the point tables with Cycles 200 to 208 and 262 to 267

The TNC interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.

Effect of the point tables with Cycles 210 to 215

The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0.

Effect of the point tables with Cycles 251 to 254

The TNC interprets the points of the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.

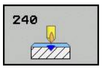
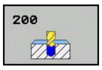
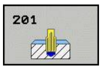
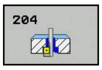
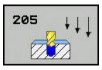
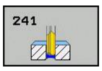
3

Fixed Cycles: Drilling

3.1 Fundamentals

Overview

The TNC offers the following cycles for all types of drilling operations:

Cycle	Soft key	Page
240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth		75
200 DRILLING With automatic pre-positioning, 2nd set-up clearance		77
201 REAMING With automatic pre-positioning, 2nd set-up clearance		79
202 BORING With automatic pre-positioning, 2nd set-up clearance		81
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing		84
204 BACK BORING With automatic pre-positioning, 2nd set-up clearance		87
205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance		90
208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance		94
241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition		97

3.2 CENTERING (Cycle 240, DIN/ISO: G240)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2 The tool is centered at the programmed feed rate **F** to the programmed centering diameter or centering depth.
3 If defined, the tool remains at the centering depth.
4 Finally, the tool path is retraced to setup clearance or—if programmed—to the 2nd setup clearance at rapid traverse **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

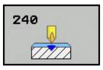
The algebraic sign for the cycle parameter **Q344** (diameter) or **Q201** (depth) determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

Danger of collision!

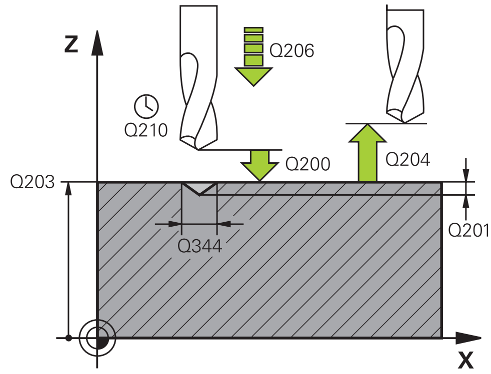
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

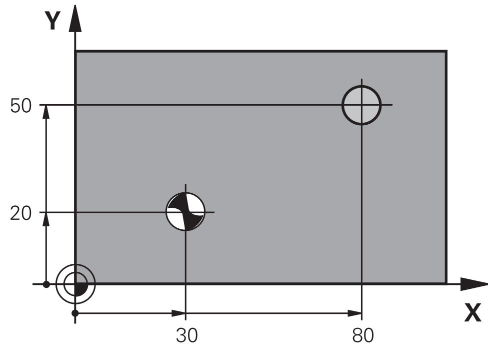
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive diameter or depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- **Select depth/diameter (0/1)** Q343: Select whether centering is based on the entered diameter or depth. If the TNC is to center based on the entered diameter, the point angle of the tool must be defined in the **T ANGLE** column of the tool table TOOL.T.
  0: Centering based on the entered depth
  1: Centering based on the entered diameter
- **Depth** Q201 (incremental): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range -99999.9999 to 99999.9999
- **Diameter (algebraic sign)** Q344: Centering diameter. Only effective if Q343=1 is defined. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during centering in mm/min. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





NC blocks

10 L Z+100 R0 FMAX
11 CYCL DEF 240 CENTERING
Q200=2 ;SET-UP CLEARANCE
Q343=1 ;SELECT DEPTH/DIA.
Q201=+0 ;DEPTH
Q344=-9 ;DIAMETER
Q206=250 ;FEED RATE FOR PLNGNG
Q211=0.1 ;DWELL TIME AT BOTTOM
Q203=+20 ;SURFACE COORDINATE
Q204=100 ;2ND SET-UP CLEARANCE
12 L X+30 Y+20 R0 FMAX M3 M99
13 L X+80 Y+50 R0 FMAX M99

3.3 DRILLING (Cycle 200)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate **F**.
3 The TNC returns the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth.
4 The tool then drills deeper by the plunging depth at the programmed feed rate F.
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6 Finally, the tool path is retraced to setup clearance from the hole bottom or—if programmed—to the 2nd setup clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

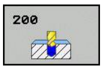
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Danger of collision!

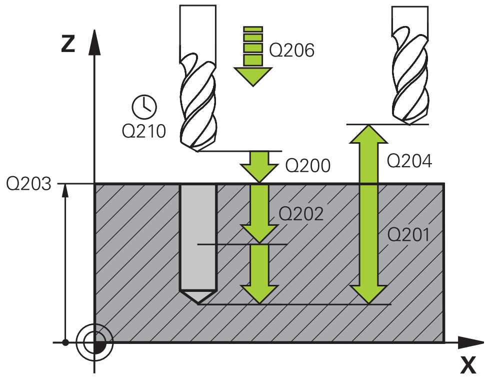
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

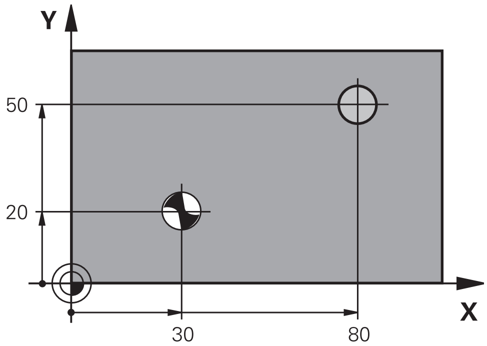
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Depth reference** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T ANGLE column of the tool table TOOL.T.
  0 = Depth referenced to the tool tip
  1 = Depth referenced to the cylindrical part of the tool





NC blocks

```
11 CYCL DEF 200 DRILLING
   Q200=2      ;SET-UP CLEARANCE
   Q201=-15    ;DEPTH
   Q206=250    ;FEED RATE FOR
                PLNGNG
   Q202=5      ;PLUNGING DEPTH
   Q211=0      ;DWELL TIME AT TOP
   Q203=+20    ;SURFACE COORDINATE
   Q204=100    ;2ND SET-UP
                CLEARANCE
   Q211=0.1    ;DWELL TIME AT
                BOTTOM
   Q395=0      ;DEPTH REFERENCE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
```

3.4 REAMING (Cycle 201, DIN/ISO: G201)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2 The tool reams to the entered depth at the programmed feed rate **F**.
3 If programmed, the tool remains at the hole bottom for the entered dwell time.
4 The tool then retracts to set-up clearance at the feed rate F, and from there—if programmed—to the 2nd set-up clearance in **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

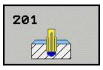
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Danger of collision!

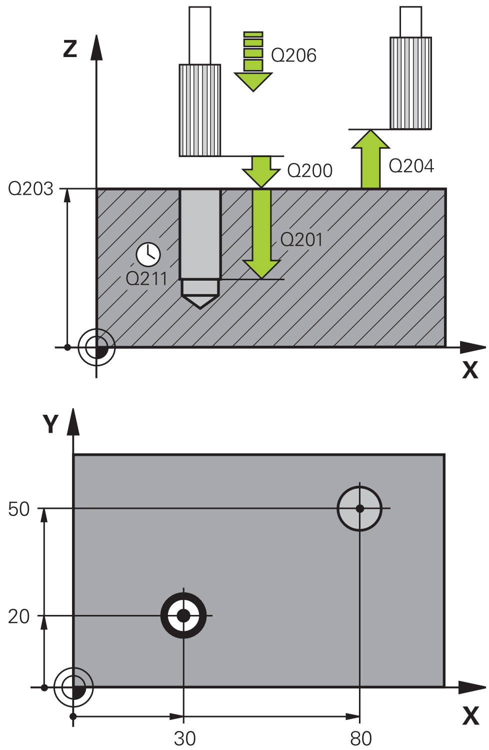
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during reaming in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the reaming feed rate. Input range 0 to 99999.999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



NC blocks

```
11 CYCL DEF 201 REAMING
   Q200=2     ;SET-UP CLEARANCE
   Q201=-15   ;DEPTH
   Q206=100   ;FEED RATE FOR
               PLNGNG
   Q211=0.5   ;DWELL TIME AT
               BOTTOM
   Q208=250   ;RETRACTION FEED
               RATE
   Q203=+20   ;SURFACE COORDINATE
   Q204=100   ;2ND SET-UP
               CLEARANCE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M9
15 L Z+100 FMAX M2
```

3.5 BORING (Cycle 202, DIN/ISO: G202)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2. The tool drills to the programmed depth at the feed rate for plunging.
3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4. The TNC then orients the spindle to the position that is defined in parameter Q336.
5. If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
6. The tool then retracts to set-up clearance at the retraction rate, and from there—if programmed—to the 2nd set-up clearance at **FMAX**. If Q214=0 the tool point remains on the wall of the hole.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

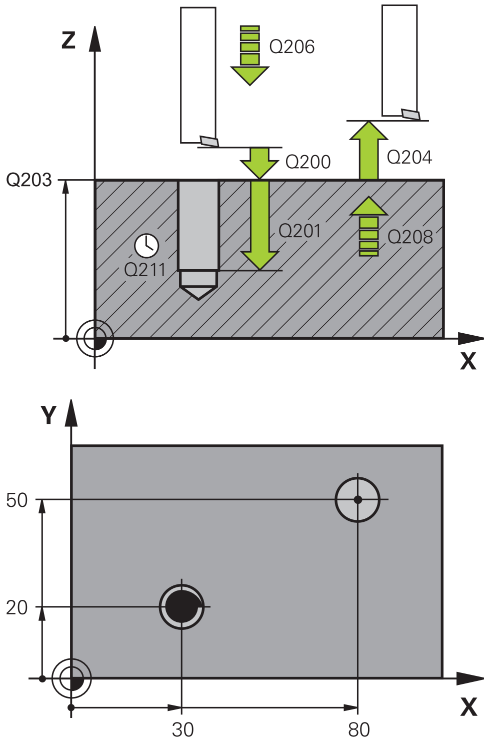
Select a disengaging direction in which the tool moves away from the edge of the hole.

Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the **Positioning with Manual Data Input** mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

During retraction the TNC automatically takes an active rotation of the coordinate system into account.

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during boring at mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at feed rate for plunging. Input range 0 to 99999.999, alternatively **FMAX**, **FAUTO**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.999
- **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the TNC retracts the
  tool on the hole bottom (after spindle orientation)
  0: Do not retract the tool
  1: Retract the tool in minus direction of the principle axis
  2: Retract the tool in minus direction of the minor axis
  3: Retract the tool in plus direction of the principle axis
  4: Retract the tool in plus direction of the minor axis
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before retracting it. Input range -360.000 to 360.000




```
10 L Z+100 R0 FMAX
11 CYCL DEF 202 BORING
   Q200=2      ;SET-UP CLEARANCE
   Q201=-15    ;DEPTH
   Q206=100    ;FEED RATE FOR
                PLNGNG
   Q211=0.5    ;DWELL TIME AT
                BOTTOM
   Q208=250    ;RETRACTION FEED
                RATE
   Q203=+20    ;SURFACE COORDINATE
   Q204=100    ;2ND SET-UP
                CLEARANCE
   Q214=1      ;DISENGAGING DIRECTN
   Q336=0      ;ANGLE OF SPINDLE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
```

3.6 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool drills to the first plunging depth at the entered feed rate **F**.
3. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at the retraction feed rate to the set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at **FMAX** to the set-up clearance above the first PLUNGING DEPTH.
4. The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5. The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

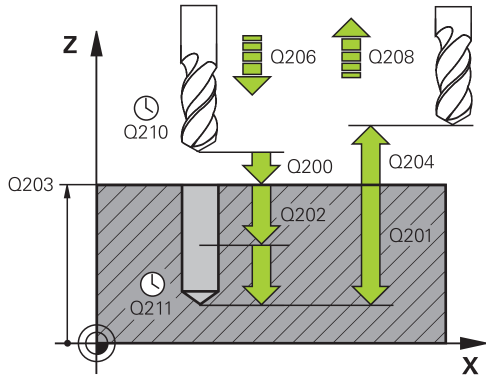
Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth and no chip breaking is defined
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999
- **No. Breaks before retracting** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip removal. For chip breaking, the TNC retracts the tool each time by the value in Q256. Input range 0 to 99999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999



NC blocks

11 CYCL DEF 203 UNIVERSAL DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.2	;DECREMENT
Q213=3	;CHIP BREAKING
Q205=3	;MIN. PLUNGING DEPTH
Q211=0.25	;DWELL TIME AT BOTTOM
Q208=500	;RETRACTION FEED RATE
Q256=0.2	;DIST. FOR CHIP BRKNG
Q395=0	;DEPTH REFERENCE

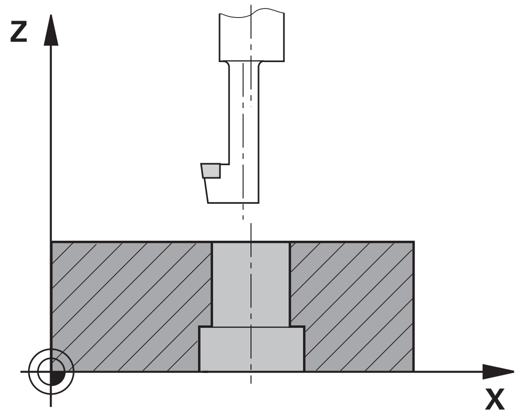
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.999, alternatively **FMAX**, **FAUTO**
- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Depth reference** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T ANGLE column of the tool table TOOL.T.
  0 = Depth referenced to the tool tip
  1 = Depth referenced to the cylindrical part of the tool

3.7 BACK BORING (Cycle 204, DIN/ISO: G204)

Cycle run

This cycle allows holes to be bored from the underside of the workpiece.

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2 The TNC then orients the spindle to the 0° position with an oriented spindle stop and displaces the tool by the off-center distance.
3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached set-up clearance on the underside of the workpiece.
4 The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.
5 If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. The TNC carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.
6 The tool then retracts to set-up clearance at the feed rate for pre-positioning, and from there—if programmed—to the 2nd set-up clearance at **FMAX**.



Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Special boring bars for upward cutting are required for this cycle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

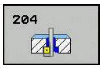
The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

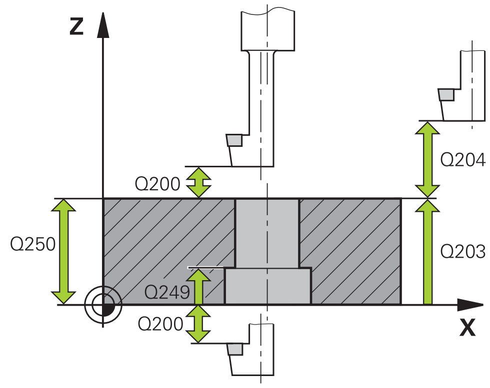
Danger of collision!

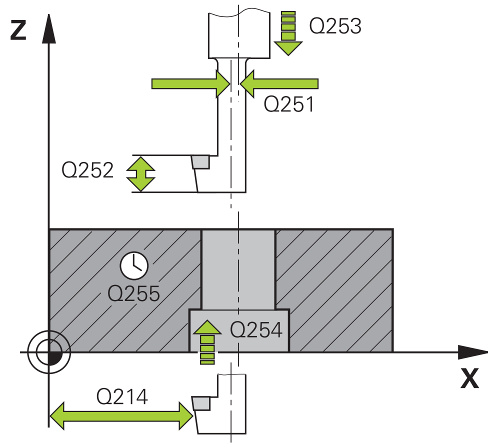
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in **Q336** (for example, in the **Positioning with Manual Data Input** mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth of counterbore** Q249 (incremental): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range -99999.9999 to 99999.9999
- **Material thickness** Q250 (incremental): Thickness of the workpiece. Input range 0.0001 to 99999.9999
- **Off-center distance** Q251 (incremental): Off-center distance for the boring bar; value from tool data sheet. Input range 0.0001 to 99999.9999
- **Tool edge height** Q252 (incremental): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet. Input range 0.0001 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- **Feed rate for back boring** Q254: Traversing speed of the tool during back boring in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time** Q255: Dwell time in seconds at the top of the bore hole. Input range 0 to 3600.000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Disengaging direction (1/2/3/4)** Q214: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation); programming 0 is not allowed
  1: Retract the tool in minus direction of the principle axis
  2: Retract the tool in minus direction of the minor axis
  3: Retract the tool in plus direction of the principle axis
  4: Retract the tool in plus direction of the minor axis
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole. Input range -360.0000 to 360.0000





NC blocks

11 CYCL DEF 204 BACK BORING	
Q200=2	;SET-UP CLEARANCE
Q249=+5	;DEPTH OF COUNTERBORE
Q250=20	;MATERIAL THICKNESS
Q251=3.5	;OFF-CENTER DISTANCE
Q252=15	;TOOL EDGE HEIGHT
Q253=750	;F PRE-POSITIONING
Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

3.8 UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. If you enter a deepened starting point, the TNC move at the defined positioning feed rate to the set-up clearance above the deepened starting point.
3. The tool drills to the first plunging depth at the entered feed rate **F**.
4. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered starting position above the first plunging depth.
5. The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
6. The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
7. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you enter different advance stop distances for **Q258** and **Q259**, the TNC will change the advance stop distances between the first and last plunging depths at the same rate.

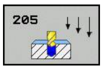
If you use **Q379** to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. Retraction movements are not changed by the TNC, therefore they are calculated with respect to the coordinate of the workpiece surface.

Danger of collision!

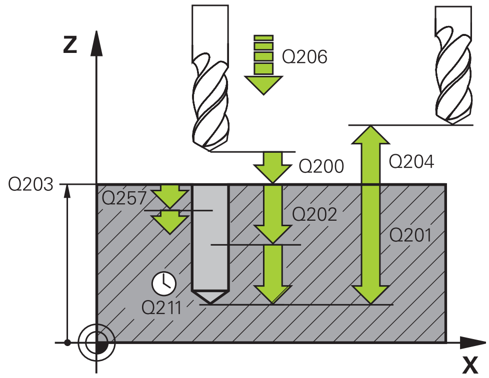
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202. Input range 0 to 99999.9999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999
- **Upper advanced stop distance** Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth. Input range 0 to 99999.9999
- **Lower advanced stop distance** Q259 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth. Input range 0 to 99999.9999
- **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000



NC blocks

11 CYCL DEF 205 UNIVERSAL PECKING	
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=15	;PLUNGING DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.5	;DECREMENT
Q205=3	;MIN. PLUNGING DEPTH
Q258=0.5	;UPPER ADV. STOP DIST.
Q259=1	;LOWER ADV. STOP DIST.
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=0.2	;DIST. FOR CHIP BRKNG
Q211=0.25	;DWELL TIME AT BOTTOM
Q379=7.5	;START POINT
Q253=750	;F PRE-POSITIONING
Q208=9999	;RETRACTION FEED RATE
Q395=0	;DEPTH REFERENCE

- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position for actual drilling operation. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance above the workpiece surface to the set-up clearance above the deepened starting point. Input range 0 to 99999.9999
- **Feed rate for pre-positioning** Q253: Defines the traversing speed of the tool when returning to the plunging depth after having retracted for chip breaking (Q256). This feed rate is also effective when the tool is positioned to a deepened starting point (Q379 not equal to 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.9999, alternatively **FMAX,FAUTO**
- **Depth reference** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T ANGLE column of the tool table TOOL.T.
  0 = Depth referenced to the tool tip
  1 = Depth referenced to the cylindrical part of the tool

3.9 BORE MILLING (Cycle 208)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface and then moves the tool to the bore hole circumference on a rounded arc (if enough space is available).
2. The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate **F**.
3. When the drilling depth is reached, the TNC once again traverses a full circle to remove the material remaining after the initial plunge.
4. The TNC then positions the tool at the center of the hole again.
5. Finally the TNC returns to the setup clearance at **FMAX**. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

An active mirror function **does not** influence the type of milling defined in the cycle.

Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the **ANGLE** column of the tool table. The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

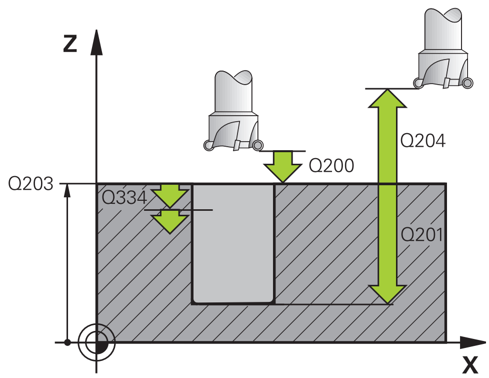
Danger of collision!

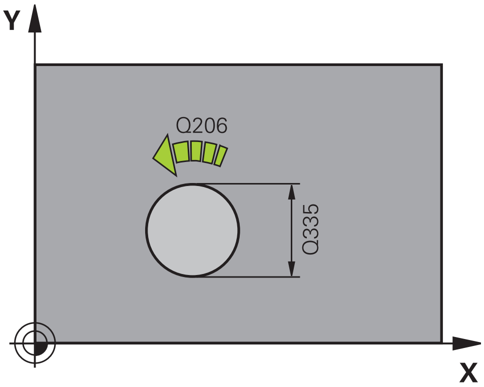
Use the machine parameter displayDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool lower edge and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during helical drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**, **FZ**
- **Infeed per helix** Q334 (incremental): Depth of the tool plunge with each helix (=360°). Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Nominal diameter** Q335 (absolute value): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation. Input range 0 to 99999.9999
- **Roughing diameter** Q342 (absolute): As soon as you enter a value greater than 0 in Q342, the TNC no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter. Input range 0 to 99999.9999
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  –1 = Up-cut





NC blocks

12 CYCL DEF 208 BORE MILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q334=1.5	;PLUNGING DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q335=25	;NOMINAL DIAMETER
Q342=0	;ROUGHING DIAMETER
Q351=+1	;CLIMB OR UP-CUT

3.10 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. Then the TNC moves the tool at the defined positioning feed rate to the set-up clearance above the deepened starting point and activates the drilling speed (**M3**) and the coolant. The TNC executes the approach motion with the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.
3. The tool drills to the hole depth at the feed rate **F**, or to the plunging depth if a smaller infeed value has been entered. The plunging depth is decreased after each infeed by the decrement. If you have entered a dwell depth, the TNC reduces the feed rate by the feed rate factor after the dwell depth has been reached.
4. If programmed, the tool remains at the hole bottom for chip breaking.
5. The TNC repeats this process (3 to 4) until the programmed total hole depth is reached.
6. After the TNC has reached the hole depth, the TNC switches off the coolant and resets the drilling speed to the value defined for retraction.
7. The tool is retracted to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

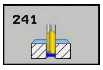
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Danger of collision!

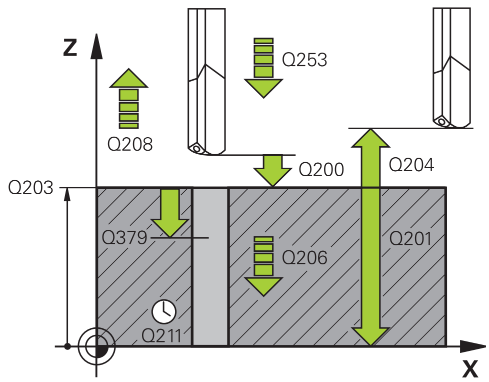
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position for actual drilling operation. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance above the workpiece surface to the set-up clearance above the deepened starting point. Input range 0 to 99999.9999
- **Feed rate for pre-positioning** Q253: Defines the traversing speed of the tool when returning to the plunging depth after having retracted for chip breaking (Q256). This feed rate is also effective when the tool is positioned to a deepened starting point (Q379 not equal to 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q206. Input range 0 to 99999.999, alternatively **FMAX**, **FAUTO**
- **Rotat. dir. of entry/exit (3/4/5)** Q426: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input:
  3: Turn the spindle with M3
  4: Turn the spindle with M4
  5: Move with stationary spindle
- **Spindle speed of entry/exit** Q427: Desired spindle speed when tool moves into and retracts from the hole. Input range 0 to 99999
- **Drilling speed** Q428: Desired speed for drilling. Input range 0 to 99999



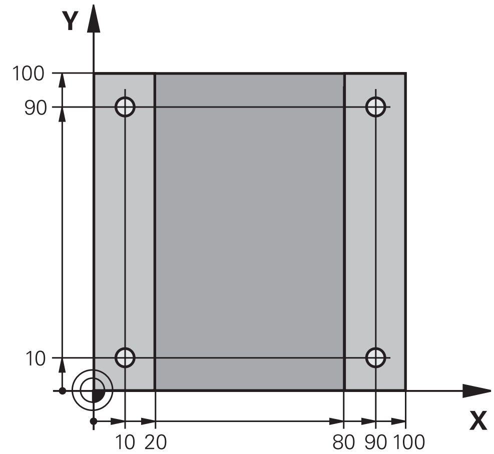
NC blocks

```
11 CYCL DEF 241 SINGLE-LIP
   D.H.DRLNG
    Q200=2     ;SET-UP CLEARANCE
    Q201=-80   ;DEPTH
    Q206=150   ;FEED RATE FOR
                PLNGNG
    Q211=0.25  ;DWELL TIME AT
                BOTTOM
    Q203=+100;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP
                CLEARANCE
    Q379=7.5   ;START POINT
    Q253=750   ;F PRE-POSITIONING
    Q208=1000;RETRACTION FEED
                RATE
    Q426=3     ;DIR. OF SPINDLE ROT.
    Q427=25    ;ROT. SPEED INFEED/
                OUT
    Q428=500   ;DRILLING SPEED
    Q429=8     ;COOLANT ON
    Q430=9     ;COOLANT OFF
    Q435=0     ;DWELL DEPTH
    Q401=100   ;FEED RATE FACTOR
    Q202=9999;MAX. PLUNGING
                DEPTH PLUNGING
                DEPTH
    Q212=0     ;DECREMENT
    Q205=0     ;MIN. PLUNGING DEPTH
                PLUNGING DEPTH
```

- **M function for coolant on?** Q429: M function for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at the deepened starting point. Input range 0 to 999
- **M function for coolant off?** Q430: M function for switching off the coolant. The TNC switches the coolant off if the tool is at the hole depth. Input range 0 to 999
- **Dwell depth** Q435 (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (standard setting). Application: During machining of through-holes some tools require a short dwell time before exiting the bottom of the hole in order to transport the chips to the top. Define a value smaller than the hole depth Q201; input range 0 to 99999.9999.
- **Feed rate factor** Q401: Factor by which the TNC reduces the feed rate after the dwell depth has been reached. Input range 0 to 100
- **Plunging depth** Q202 (incremental): Infeed per cut. The depth does not have to be a multiple of the plunging depth. Input range 0 to 99999.9999
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999

3.11 Programming Examples

Example: Drilling cycles



0 BEGIN PGM C200 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 S4500	Tool call (tool radius 3)
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT BOTTOM	
Q395=0 ;DEPTH REFERENCE	
6 L X+10 Y+10 R0 FMAX M3	Approach hole 1, spindle ON
7 CYCL CALL	Cycle call
8 L Y+90 R0 FMAX M99	Approach hole 2, call cycle
9 L X+90 R0 FMAX M99	Approach hole 3, call cycle
10 L Y+10 R0 FMAX M99	Approach hole 4, call cycle
11 L Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM C200 MM	

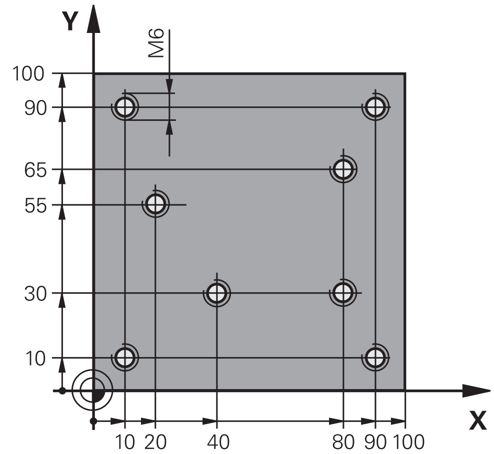
Example: Using drilling cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the pattern definition PATTERN DEF POS and are called by the TNC with CYCL CALL PAT.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering (tool radius 4)
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)



0 BEGIN PGM 1 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 Y+0	
3 TOOL CALL 1 Z S5000	Call the centering tool (tool radius 4)
4 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
5 PATTERN DEF	Define all drilling positions in the point pattern
POS1(X+10 Y+10 Z+0)	
POS2(X+40 Y+30 Z+0)	
POS3(X+20 Y+55 Z+0)	
POS4(X+10 Y+90 Z+0)	
POS5(X+90 Y+90 Z+0)	
POS6(X+80 Y+65 Z+0)	
POS7(X+80 Y+30 Z+0)	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q343=0 ;SELECT DEPTH/DIA.	
Q201=-2 ;DEPTH	
Q344=-10 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT BOTTOM	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
7 CYCL CALL PAT F5000 M13	Call the cycle in connection with the hole pattern
8 L Z+100 R0 FMAX	Retract the tool, change the tool
9 TOOL CALL 2 Z S5000	Call the drilling tool (radius 2.4)
10 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)

11 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT BOTTOM	
Q395=0 ;DEPTH REFERENCE	
12 CYCL CALL PAT F5000 M13	Call the cycle in connection with the hole pattern
13 L Z+100 R0 FMAX	Retract the tool
14 TOOL CALL 3 Z S200	Call the tapping tool (radius 3)
15 L Z+50 R0 FMAX	Move tool to clearance height
16 CYCL DEF 206 TAPPING NEW	Cycle definition for tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;THREAD DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT BOTTOM	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
17 CYCL CALL PAT F5000 M13	Call the cycle in connection with the hole pattern
18 L Z+100 R0 FMAX M2	Retract the tool, end program
19 END PGM 1 MM	

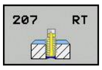
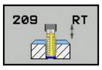
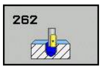
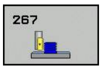
4

Fixed Cycles: Tapping / Thread Milling

4.1 Fundamentals

Overview

The TNC offers the following cycles for all types of threading operations:

Cycle	Soft key	Page
206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		105
207 TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		108
209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking		111
262 THREAD MILLING Cycle for milling a thread in pre-drilled material		117
263 THREAD MILLING/ COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer		120
264 THREAD DRILLING/MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool		124
265 HELICAL THREAD DRILLING/ MILLING Cycle for milling the thread into solid material		128
267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer		132

4.2 TAPPING with a floating tap holder (Cycle 206, DIN/ISO: G206)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
4. At the set-up clearance, the direction of spindle rotation reverses once again.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match. In Cycle 206 the TNC uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

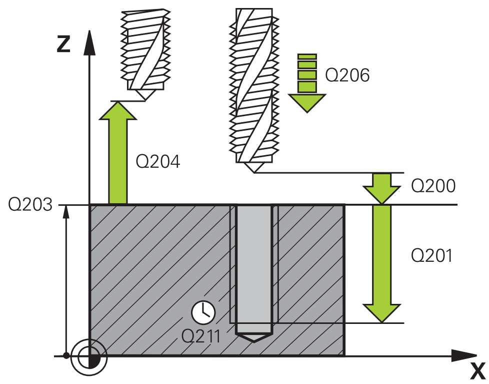
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

Guide value: 4x pitch.

- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Feed rate F** Q206: Traversing speed of the tool during tapping. Input range 0 to 99999.999 alternatively **FAUTO**
- **Dwell time at bottom** Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



NC blocks

25 CYCL DEF 206 TAPPING NEW	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;THREAD DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q211=0.25	;DWELL TIME AT BOTTOM
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

The feed rate is calculated as follows: F = S x p

F: Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.

4.3 RIGID TAPPING without a floating tap holder (Cycle 207, DIN/ ISO: G207)

Cycle run

The TNC cuts the thread without a floating tap holder in one or more passes.

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
4. The TNC stops the spindle turning at set-up clearance.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The TNC calculates the feed rate from the spindle speed. If the feed-rate override is used during tapping, the TNC automatically adjusts the feed rate.

The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

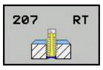
If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

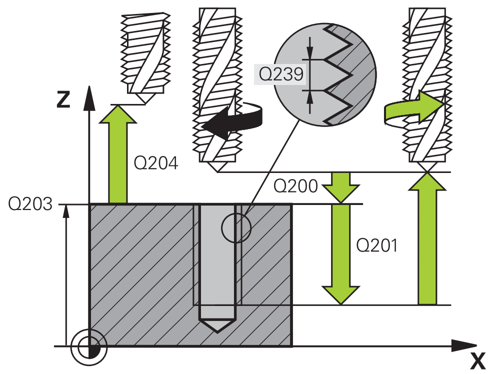
Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



NC blocks

26 CYCL DEF 207 RIGID TAPPING NEW	
Q200=2	**;SET-UP CLEARANCE**
Q201=-20	**;THREAD DEPTH**
Q239=+1	**;THREAD PITCH**
Q203=+25	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**

Retracting after a program interruption

Retracting in the Manual Operation mode

You can interrupt the thread cutting process by pressing the NC Stop key. A soft key for retracting the tool from the thread is displayed in the soft-key row below the screen. When you press this soft key and the NC Start key, the tool retracts from the hole and returns to the starting point of machining. The spindle is stopped automatically and the TNC displays a message.

Retracting in the Program Run, Single Block or Full Sequence mode

You can interrupt the thread cutting process by pressing the NC Stop key and then INTERNAL STOP. The TNC displays the **MANUAL OPERATION** soft key. After pressing **MANUAL OPERATION**, you can retract the tool in the active spindle axis. To resume machining after the interruption, press the **RESTORE POSITION** soft key and NC Start. The TNC moves the tool back to the starting position.

When retracting the tool you can move it in the positive and negative tool axis directions. Please keep this in mind during retraction—danger of collision!

4.4 TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209)

Cycle run

The TNC machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface. There it carries out an oriented spindle stop.
2 The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition. If you have defined a factor for increasing the spindle speed, the TNC retracts from the hole at the corresponding speed.
3 It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4 The TNC repeats this process (2 to 3) until the programmed thread depth is reached.
5 The tool is then retracted to set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
6 The TNC stops the spindle turning at set-up clearance.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter "thread depth" determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the feed-rate override is used during tapping, the TNC automatically adjusts the feed rate.

The feed-rate override knob is disabled.

If you defined an rpm factor for fast retraction in cycle parameter **Q403**, the TNC limits the speed to the maximum speed of the active gear range.

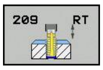
At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

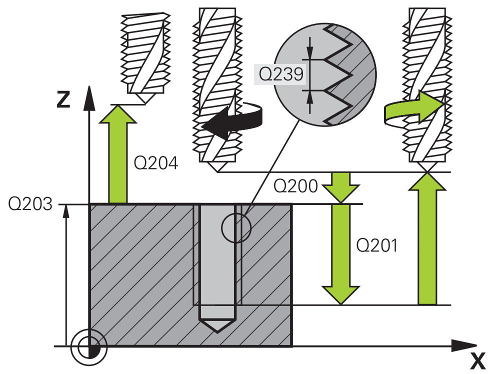
Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256: The TNC multiplies the pitch Q239 by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter Q256 = 0, the TNC retracts the tool completely from the hole (to the set-up clearance) for chip breaking. Input range 0.000 to 99999.999
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before machining the thread. This allows you to regroove the thread, if required. Input range -360.0000 to 360.0000
- **RPM factor for retraction** Q403: Factor by which the TNC increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole. Input range 0.0001 to 10. Maximum increase to maximum speed of the active gear range.



NC blocks

26 CYCL DEF 209 TAPPING W/ CHIP BRKG	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q239=+1	;THREAD PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=+25	;DIST. FOR CHIP BRKNG
Q336=50	;ANGLE OF SPINDLE
Q403=1.5	;RPM FACTOR

Retracting after a program interruption

Retracting in the Manual Operation mode

You can interrupt the thread cutting process by pressing the NC Stop key. A soft key for retracting the tool from the thread is displayed in the soft-key row below the screen. When you press this soft key and the NC Start key, the tool retracts from the hole and returns to the starting point of machining. The spindle is stopped automatically and the TNC displays a message.

Retracting in the Program Run, Single Block or Full Sequence mode

You can interrupt the thread cutting process by pressing the NC Stop key and then INTERNAL STOP. The TNC displays the **MANUAL OPERATION** soft key. After pressing **MANUAL OPERATION**, you can retract the tool in the active spindle axis. To resume machining after the interruption, press the **RESTORE POSITION** soft key and NC Start. The TNC moves the tool back to the starting position.

> When retracting the tool you can move it in the positive and negative tool axis directions. Please keep this in mind during retraction—danger of collision!

4.5 Fundamentals of Thread Milling

Prerequisites

- Your machine tool should feature internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars).
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius **DR** in the **TOOL CALL**.
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread / – = left-hand thread) and milling method Q351 (+1 = climb / –1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/ Up-cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	–	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	–	+1(RL)	Z–

External thread	Pitch	Climb/ Up-cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	–	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	–	+1(RL)	Z+

The TNC references the programmed feed rate during thread milling to the tool cutting edge. Since the TNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRROR IMAGE in only one axis.

Danger of collision!

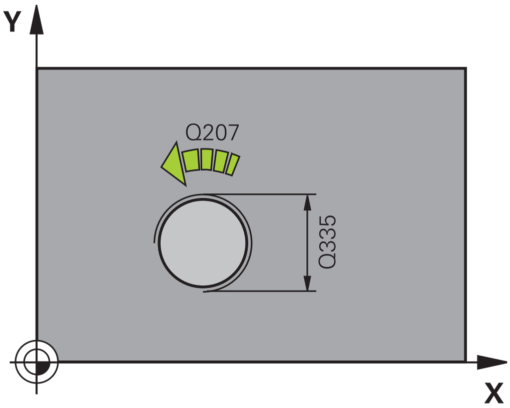
Always program the same algebraic sign for the infeeds: Cycles comprise several sequences of operation that are independent of each other.
The order of precedence according to which the work direction is determined is described with the individual cycles. For example, if you only want to repeat the countersinking process of a cycle, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

Procedure in case of a tool break

If a tool break occurs during thread cutting, stop program run, change to the Positioning with MDI operating mode and move the tool on a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

4.6 THREAD MILLING (Cycle 262, DIN/ISO: G262)

Cycle run



1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3. The tool then moves tangentially on a helical path to the thread major diameter. Before the helical approach, a compensating motion of the tool axis is carried out in order to begin at the programmed starting plane for the thread path.
4. Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
5. After that the tool departs the contour tangentially and returns to the starting point in the working plane.
6. At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter "thread depth" determines the working direction.

If you program the thread DEPTH = 0, the cycle will not be executed.

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the pitch of the tool diameter is four times smaller than the nominal thread diameter.

Note that the TNC makes a compensation movement in the tool axis before the approach movement. The length of the compensation movement is at most half of the thread pitch. Ensure sufficient space in the hole!

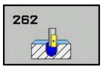
If you change the thread depth, the TNC automatically changes the starting point for the helical movement.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

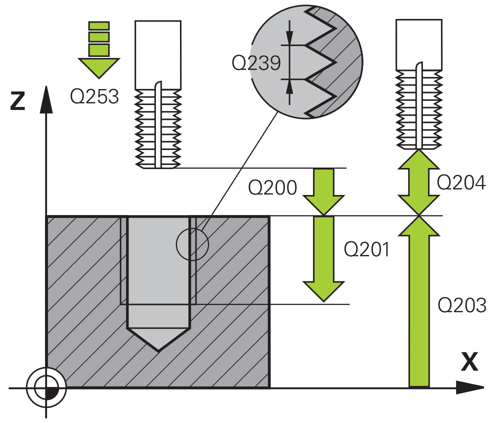
Cycle parameters

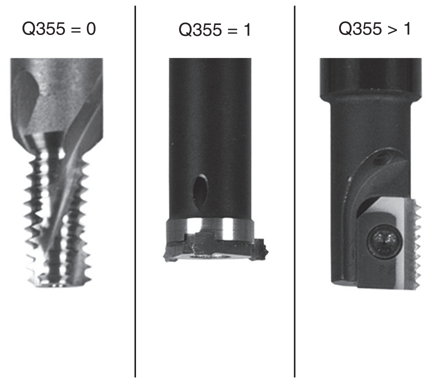


- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Threads per step** Q355: Number of thread starts by which the tool is displaced:
  0 = one helix on the thread depth
  1 = continuous helix on the complete thread length
  >1 = several helix paths with approach and departure, between these the TNC sets the tool by Q355 x pitch. Input range 0 to 99999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  –1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Feed rate for approaching** Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.999 alternatively **FAUTO**





NC blocks

25 CYCL DEF 262 THREAD MILLING	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-20	;THREAD DEPTH
Q355=0	;THREADS PER STEP
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q207=500	;FEED RATE FOR MILLING
Q512=0	;FEED RATE FOR APPROACHING

4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO:G263)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Countersinking

2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up clearance, and then at the feed rate for countersinking to the countersinking depth.

3 If a safety clearance to the side has been entered, the TNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth.

4 Then, depending on the available space, the TNC makes a tangential approach to the core diameter, either tangentially from the center or with a pre-positioning move to the side, and follows a circular path.

Countersinking at front

5 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

6 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

7 The tool then moves in a semicircle to the hole center.

Thread milling

8 The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).

9 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.

10 After that the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Countersinking depth
3. Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you want to countersink at front, define the countersinking depth as 0.

Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

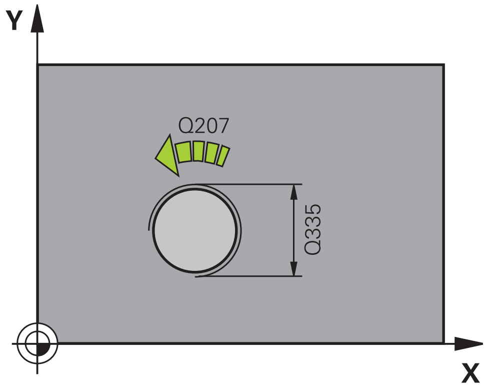
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

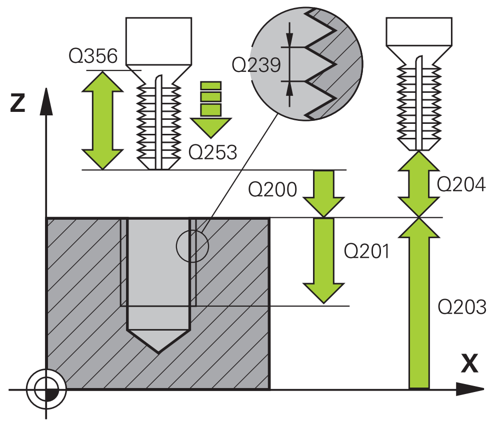
Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

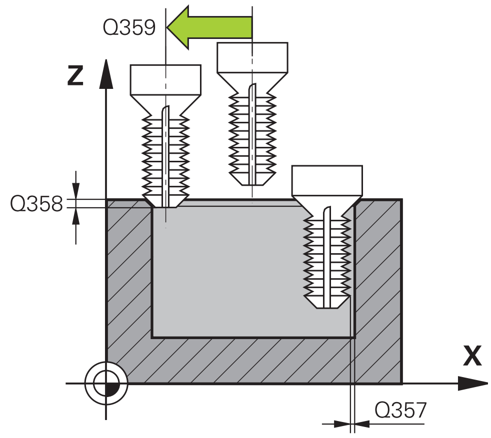
  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Countersinking depth** Q356 (incremental): Distance between tool tip and the top surface of the workpiece. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  –1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Set-up clearance to the side** Q357 (incremental): Distance between tool tooth and the wall of the hole. Input range 0 to 99999.9999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999





- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Feed rate for countersinking** Q254: Traversing speed of the tool during countersinking in mm/min. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Feed rate for approaching** Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.999 alternatively **FAUTO**



NC blocks

25 CYCL DEF 263 THREAD MLLNG/ CNTSNKG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;THREAD DEPTH
Q356=-20	;COUNTERSINKING DEPTH
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q357=0.2	;CLEARANCE TO SIDE
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLING
Q512=0	;FEED RATE FOR APPROACHING

4.8 THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Drilling

2 The tool drills to the first plunging depth at the programmed feed rate for plunging.

3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered starting position above the first plunging depth.

4 The tool then advances with another infeed at the programmed feed rate.

5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

7 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

8 The tool then moves in a semicircle to the hole center.

Thread milling

9 The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).

10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.

11 After that the tool departs the contour tangentially and returns to the starting point in the working plane.

12 At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Countersinking depth
3. Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters

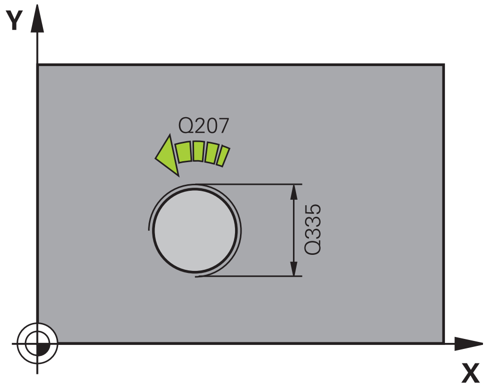
- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

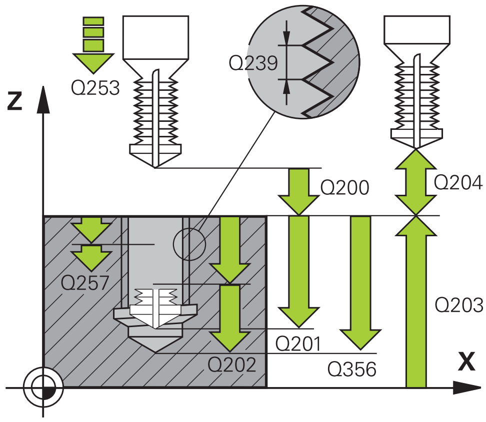
  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Total hole depth** Q356 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  –1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Plunging depth** Q202 (incremental): Infeed per cut. The depth does not have to be a multiple of the plunging depth. Input range 0 to 99999.9999

The TNC will go to depth in one movement if:

- the plunging depth is equal to the depth
- the plunging depth is greater than the depth





NC blocks

25 CYCL DEF 264 THREAD DRILLNG/ MLLNG
Q335=10 ;NOMINAL DIAMETER

- **Upper advanced stop distance** Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole. Input range 0 to 99999.9999
- **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool when moving into the workpiece in mm/min. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Feed rate for approaching** Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.999 alternatively **FAUTO**

Q239=+1.5	;THREAD PITCH
Q201=-16	;THREAD DEPTH
Q356=-20	;TOTAL HOLE DEPTH
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q202=5	;PLUNGING DEPTH
Q258=0.2	;ADV. STOP DIST.
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=0.2	;DIST. FOR CHIP BRKNG
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q206=150	;FEED RATE FOR PLNGNG
Q207=500	;FEED RATE FOR MILLING
Q512=0	;FEED RATE FOR APPROACHING

4.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Countersinking at front

2. If countersinking occurs before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking occurs after thread milling, the TNC moves the tool to the countersinking depth at the feed rate for pre-positioning.
3. The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
4. The tool then moves in a semicircle to the hole center.

Thread milling

5. The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.
6. The tool then approaches the thread diameter tangentially in a helical movement.
7. The tool moves on a continuous helical downward path until it reaches the thread depth.
8. After that the tool departs the contour tangentially and returns to the starting point in the working plane.
9. At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you change the thread depth, the TNC automatically changes the starting point for the helical movement.

The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

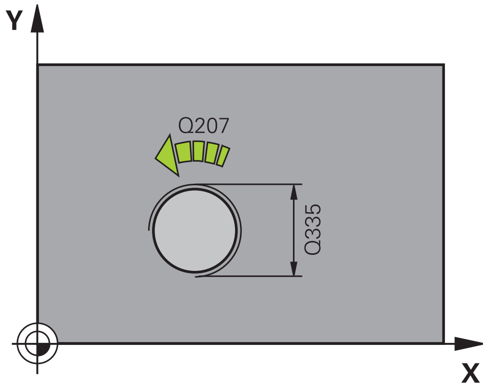
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

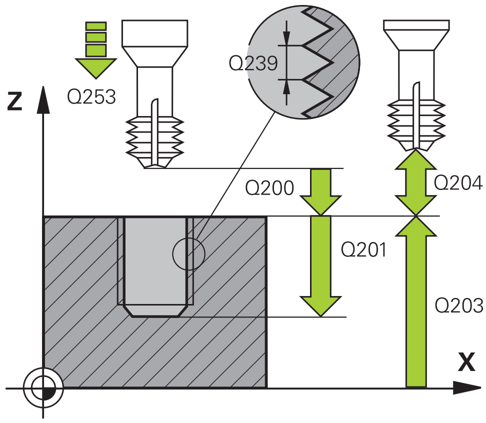
Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

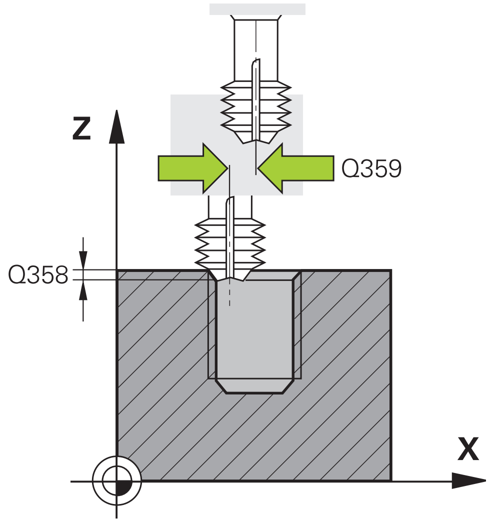
  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- **Countersinking** Q360: Running the chamfer
  0 = before thread milling
  1 = after thread milling
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999





- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Feed rate for countersinking** Q254: Traversing speed of the tool during countersinking in mm/min. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**



NC blocks

25 CYCL DEF 265 HEL. THREAD DRLG/ MLG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;THREAD DEPTH
Q253=750	;F PRE-POSITIONING
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q360=0	;COUNTERSINKING
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLING

4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Countersinking at front

2 The TNC moves on the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.

3 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

4 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

5 The tool then moves in a semicircle to the starting point.

Thread milling

6 The TNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.

7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.

8 The tool then approaches the thread diameter tangentially in a helical movement.

9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.

10 After that the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.

Please note while programming:

Program a positioning block for the starting point (stud center) in the working plane with radius compensation **R0**.

The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

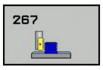
The algebraic sign for the cycle parameter "thread depth" determines the working direction.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

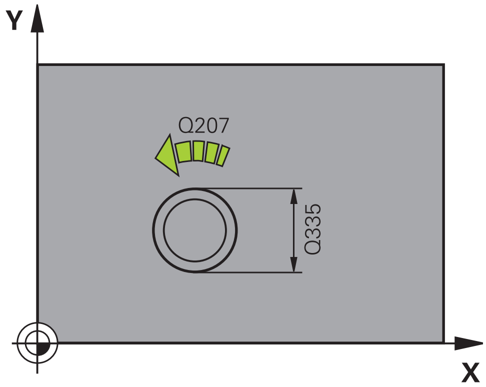
Cycle parameters

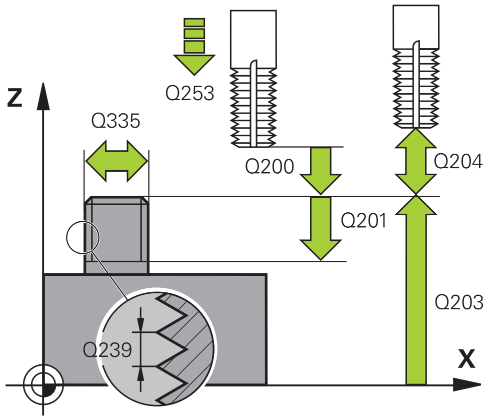


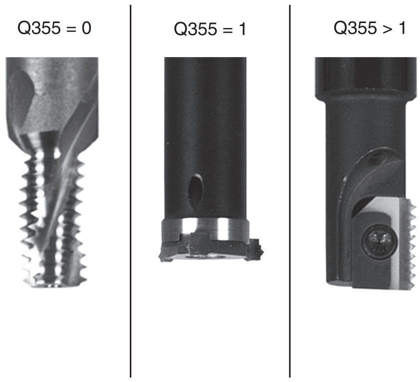
- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

  + = right-hand thread

  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Threads per step** Q355: Number of thread starts by which the tool is displaced:
  0 = one helix on the thread depth
  1 = continuous helix on the complete thread length
  >1 = several helix paths with approach and departure, between these the TNC sets the tool by Q355 x pitch. Input range 0 to 99999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  –1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999







- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Feed rate for countersinking** Q254: Traversing speed of the tool during countersinking in mm/min. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Feed rate for approaching** Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.999 alternatively **FAUTO**

NC blocks

```
25 CYCL DEF 267 OUTSIDE THREAD
   MLLNG
   Q335=10    ;NOMINAL DIAMETER
   Q239=+1.5 ;THREAD PITCH
   Q201=-20   ;THREAD DEPTH
   Q355=0     ;THREADS PER STEP
   Q253=750   ;F PRE-POSITIONING
   Q351=+1    ;CLIMB OR UP-CUT
   Q200=2     ;SET-UP CLEARANCE
   Q358=+0    ;DEPTH AT FRONT
   Q359=+0    ;OFFSET AT FRONT
   Q203=+30   ;SURFACE COORDINATE
   Q204=50    ;2ND SET-UP
              CLEARANCE
   Q254=150   ;F COUNTERBORING
   Q207=500   ;FEED RATE FOR
              MILLING
   Q512=0     ;FEED RATE FOR
              APPROACHING
```

4.11 Programming Examples

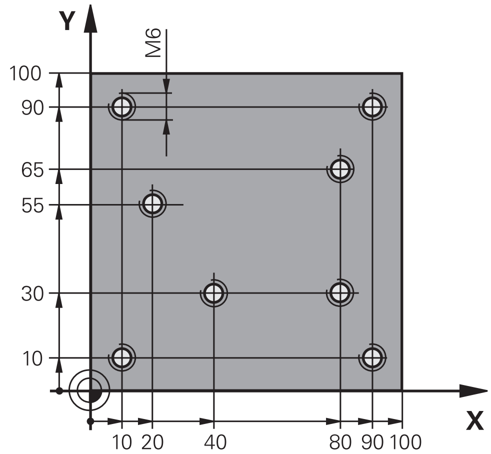
Example: Thread milling

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the TNC with **CYCL CALL PAT**.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping



0 BEGIN PGM 1 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 Y+0	
3 TOOL CALL 1 Z S5000	Call tool: centering drill
4 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
5 SEL PATTERN "TAB1"	Definition of point table
6 CYCL DEF 240 CENTERING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q343=1 ;SELECT DIA./DEPTH	
Q201=-3.5 ;DEPTH	
Q344=-7 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q11=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT, feed rate between the points: 5000 mm/min
11 L Z+100 R0 FMAX M6	Retract the tool, change the tool
12 TOOL CALL 2 Z S5000	Call tool: drill
13 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table

Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
15 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
16 L Z+100 R0 FMAX M6	Retract the tool, change the tool
17 TOOL CALL 3 Z S200	Call tool: tap
18 L Z+50 R0 FMAX	Move tool to clearance height
19 CYCL DEF 206 TAPPING	Cycle definition for tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
20 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
21 L Z+100 R0 FMAX M2	Retract the tool, end program
22 END PGM 1 MM	

Point table TAB1.PNT

TAB1. PNT MM
NR X Y Z
0 +10 +10 +0
1 +40 +30 +0
2 +90 +10 +0
3 +80 +30 +0
4 +80 +65 +0
5 +90 +90 +0
6 +10 +90 +0
7 +20 +55 +0
[END]

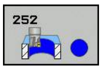
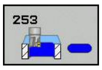
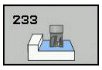
5

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.1 Fundamentals

Overview

The TNC offers the following cycles for machining pockets, studs and slots :

Cycle	Soft key	Page
251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		141
252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		145
253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal plunging		150
254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal plunging		154
256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required		159
257 CIRCULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required		163
233 FACE MILLING Machining the face with up to 3 limits		167

5.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251)

Cycle run

Use Cycle 251 RECTANGULAR POCKET to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1 The tool plunges the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter Q366.
2 The TNC roughs out the pocket from the inside out, taking the overlap factor (parameter Q370) and the finishing allowance (parameters Q368 and Q369) into account.
3 At the end of the roughing operation, the TNC moves the tool tangentially away from the pocket wall, then moves by the set-up clearance above the current plunging depth and returns from there at rapid traverse to the pocket center.
4 This process is repeated until the programmed pocket depth is reached.

Finishing

5 If finishing allowances are defined, the tool plunges the workpiece at the pocket center and moves to the plunging depth for finishing. The TNC first finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.
6 Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC returns the tool to the starting position.

At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

The TNC outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

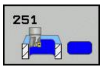
Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

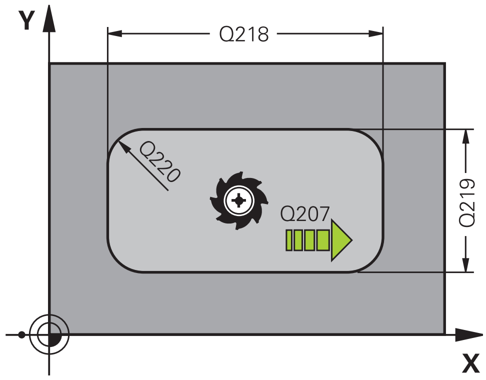
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

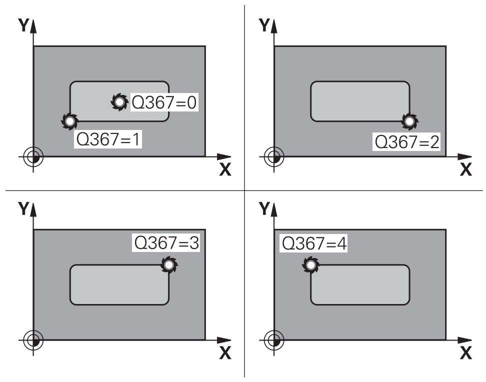
If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.

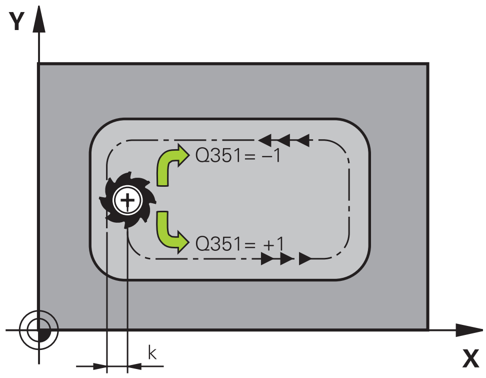
Cycle parameters

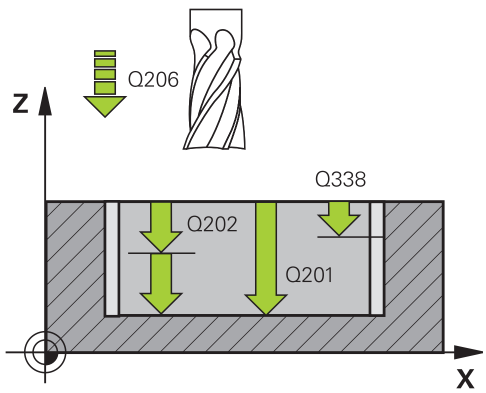


- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **1st side length** Q218 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q219 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Corner radius** Q220: Radius of the pocket corner. If you have entered 0 here, the TNC assumes that the corner radius is equal to the tool radius. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Angle of rotation** Q224 (absolute): Angle by which the entire machining is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.0000 to 360.0000
- **Pocket position** Q367: Position of the pocket in reference to the position of the tool when the cycle is called:
  0: Tool position = pocket center
  1: Tool position = left corner below
  2: Tool position = right corner below
  3: Tool position = right corner top
  4: Tool position = left top corner top
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

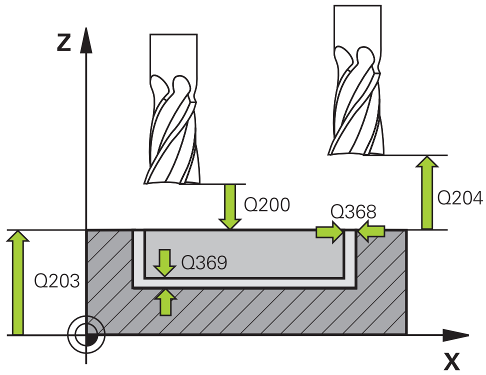








- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414 alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  0: vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table
  1: helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message
  2: reciprocal plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message. The reciprocation length depends on the plunging angle. As a minimum value the TNC uses twice the tool diameter
  PREDEF: The TNC uses the value from the GLOBAL DEF block
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**



NC blocks

8 CYCL DEF 251 RECTANGULAR POCKET	
Q215=0	;MACHINING OPERATION
Q218=80	;FIRST SIDE LENGTH
Q219=60	;2ND SIDE LENGTH
Q220=5	;CORNER RADIUS
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;POCKET POSITION
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q370=1	;TOOL PATH OVERLAP
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
9 L X+50 Y+50 R0 FMAX M3 M99	

5.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)

Cycle run

Use Cycle 252 CIRCULAR POCKET to machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1. The TNC first moves the tool at rapid traverse to the set-up clearance Q200 above the workpiece.
2. The tool plunges to the first plunging depth at the pocket center. Specify the plunging strategy with parameter Q366.
3. The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowance (Parameters Q368 and Q369) into account.
4. At the end of a roughing operation, the TNC moves the tool tangentially away from the pocket wall by the set-up clearance Q200 in the working plane, then retracts the tool by Q200 at rapid traverse and returns it from there at rapid traverse to the pocket center.
5. Steps 2 to 4 are repeated until the programmed pocket depth is reached, taking the finishing allowance Q369 into account.
6. If only roughing is programmed (Q215=1), the tool moves away from the pocket wall tangentially by the set-up clearance Q200, then retracts at rapid traverse to the 2nd set-up clearance Q200 in the tool axis and returns at rapid traverse to the pocket center.

Finishing

1 Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified.
2 The TNC positions the tool in the tool axis in front of the pocket wall, taking the finishing allowance Q368 and the set-up clearance Q200 into account.
3 The TNC clears the pocket from the inside out until the diameter Q223 is reached.
4 Then the TNC again positions the tool in the tool axis in front of the pocket wall, taking the finishing allowance Q368 and the set-up clearance Q200 into account, and repeats the finishing process of the pocket wall at the next depth.
5 The TNC repeats this process until the programmed diameter is reached.
6 After machining to the diameter Q223, the TNC retracts the tool tangentially by the finishing allowance Q368 plus the set-up clearance Q200 in the working plane, then retracts at rapid traverse to the set-up clearance Q200 in the tool axis and returns to the pocket center.
7 Next, the TNC moves the tool in the tool axis to the depth Q201 and finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.
8 The TNC repeats this process until the depth Q201 plus Q369 is reached.
9 Finally, the tool moves away from the pocket wall tangentially by the set-up clearance Q200, then retracts at rapid traverse to the set-up clearance Q200 in the tool axis and returns at rapid traverse to the pocket center.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position (circle center) with radius compensation **R0**.

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC returns the tool to the starting position.

At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

The TNC outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

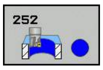
Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

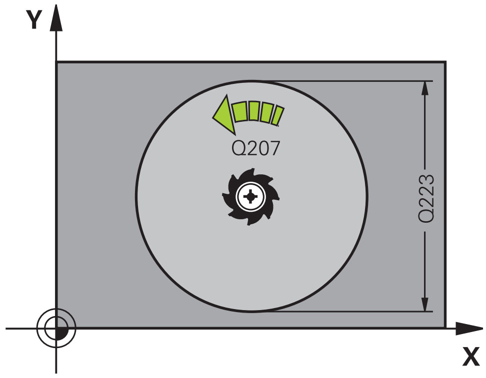
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

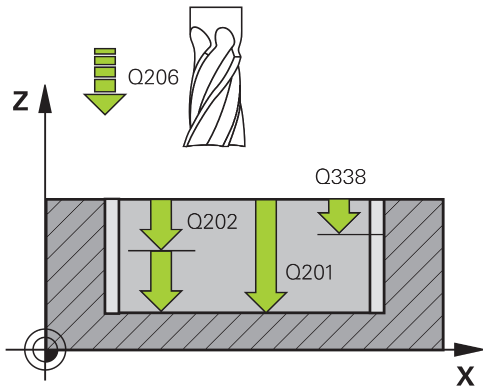
If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.

Cycle parameters

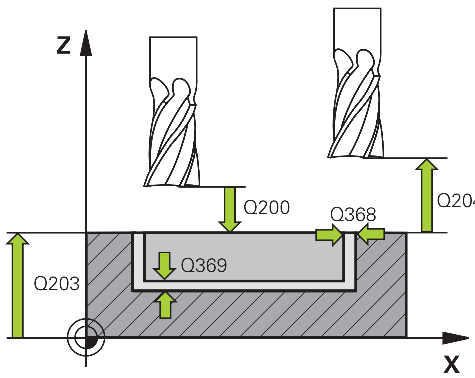


- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Circle diameter** Q223: Diameter of the finished pocket. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**





- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.9999; alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  - 0 = vertical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as 0 or 90. The TNC will otherwise display an error message.
  - 1 = helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
  - Alternative: **PREDEF**
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate reference (0...3)** Q439: Define a reference for the programmed feed rate:
  0: The feed rate refers to the center point path of the tool
  1: The feed rate refers to the tool cutting edge only during side finishing; otherwise, it refers to the center point path
  2: The feed rate refers to the tool cutting edge during side **and** floor finishing; otherwise, it refers to the center point path
  3: The feed rate always refers to the tool cutting edge



NC blocks

8 CYCL DEF 252 CIRCULAR POCKET	
Q215=0	**;MACHINING OPERATION**
Q223=60	**;CIRCLE DIAMETER**
Q368=0.2	**;ALLOWANCE FOR SIDE**
Q207=500	**;FEED RATE FOR MILLING**
Q351=+1	**;CLIMB OR UP-CUT**
Q201=-20	**;DEPTH**
Q202=5	**;PLUNGING DEPTH**
Q369=0.1	**;ALLOWANCE FOR FLOOR**
Q206=150	**;FEED RATE FOR PLNGNG**
Q338=5	**;INFEED FOR FINISHING**
Q200=2	**;SET-UP CLEARANCE**
Q203=+0	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q370=1	**;TOOL PATH OVERLAP**
Q366=1	**;PLUNGE**
Q385=500	**;FINISHING FEED RATE**
Q439=3	**;FEED RATE REFERENCE**
9 L X+50 Y+50 R0 FMAX M3 M99	

5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253)

Cycle run

Use Cycle 253 to completely machine a slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1 Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter Q366.

2 The TNC roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 und Q369) into account.

3 The TNC retracts the tool by the set-up clearance Q200. If the slot width matches the cutter diameter, the TNC retracts the tool from the slot after each infeed.

4 This process is repeated until the programmed slot depth is reached.

Finishing

5 Inasmuch as finishing allowances are defined, the TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially in the left slot arc.

6 Then the TNC finishes the floor of the slot from the inside out.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

At the end of the cycle the TNC merely moves the tool in working plane back to the center of the slot; in the other working plane axis the TNC does not do any positioning. If you define a slot position not equal to 0, then the TNC only positions the tool in the tool axis to the 2nd set-up clearance. Prior to a new cycle call, move the tool back to the starting position or program always absolute traverse motions after the cycle call.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If the slot width is greater than twice the tool diameter, the TNC roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

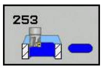
Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

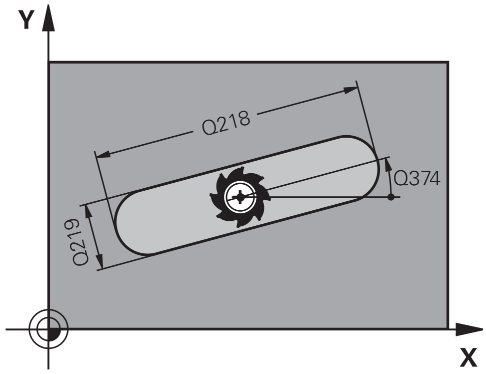
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

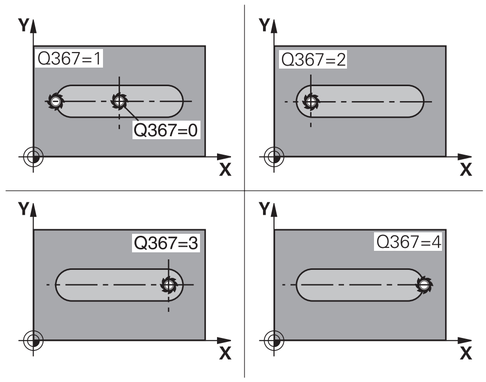
If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool to the first plunging depth at rapid traverse!

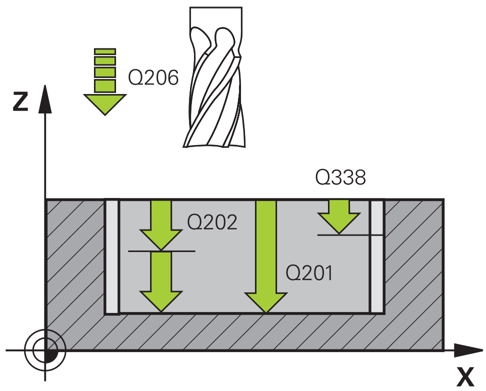
Cycle parameters

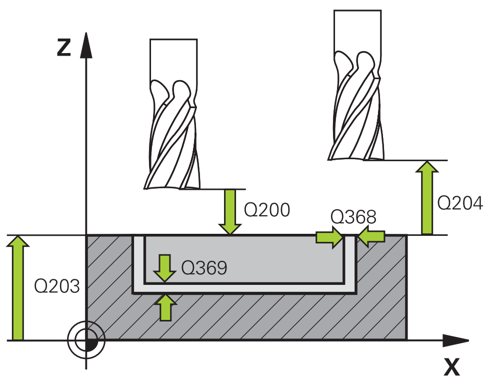


- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Slot length** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range 0 to 99999.9999
- **Slot width** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Angle of rotation** Q374 (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.000 to 360.000
- **Slot position (0/1/2/3/4)** Q367: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position = slot center
  1: Tool position = left end of slot
  2: Tool position = center of left slot arc
  3: Tool position = center of right slot arc
  4: Tool position = right end of slot
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999









- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  - 0 = vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
  - 1, 2 = reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
  - Alternative: **PREDEF**
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate reference (0...3)** Q439: Define a reference for the programmed feed rate:
  0: The feed rate refers to the center point path of the tool
  1: The feed rate refers to the tool cutting edge only during side finishing; otherwise, it refers to the center point path
  2: The feed rate refers to the tool cutting edge during side **and** floor finishing; otherwise, it refers to the center point path
  3: The feed rate always refers to the tool cutting edge

NC blocks

```
8 CYCL DEF 253 SLOT MILLING
    Q215=0     ;MACHINING OPERATION
    Q218=80    ;SLOT LENGTH
    Q219=12    ;SLOT WIDTH
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q374=+0    ;ANGLE OF ROTATION
    Q367=0     ;SLOT POSITION
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q369=0.1   ;ALLOWANCE FOR FLOOR
    Q206=150   ;FEED RATE FOR PLNGNG
    Q338=5     ;INFEED FOR FINISHING
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q366=1     ;PLUNGE
    Q385=500   ;FINISHING FEED RATE
    Q439=0     ;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99
```

5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)

Cycle run

Use Cycle 254 to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1 The tool moves in a reciprocating motion in the slot center at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter Q366.
2 The TNC roughs out the slot from the inside out, taking the finishing allowances (parameter Q368) into account.
3 The TNC retracts the tool by the set-up clearance Q200. If the slot width matches the cutter diameter, the TNC retracts the tool from the slot after each infeed.
4 This process is repeated until the programmed slot depth is reached.

Finishing

5 Inasmuch as finishing allowances are defined, the TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially.
6 Then the TNC finishes the floor of the slot from the inside out.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

At the end of the cycle the TNC returns the tool to the starting point (center of the pitch circle) in the working plane. Exception: if you define a slot position not equal to 0, then the TNC only positions the tool in the tool axis to the 2nd set-up clearance. In these cases, always program absolute traverse movements after the cycle call.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If the slot width is greater than twice the tool diameter, the TNC roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

Danger of collision!

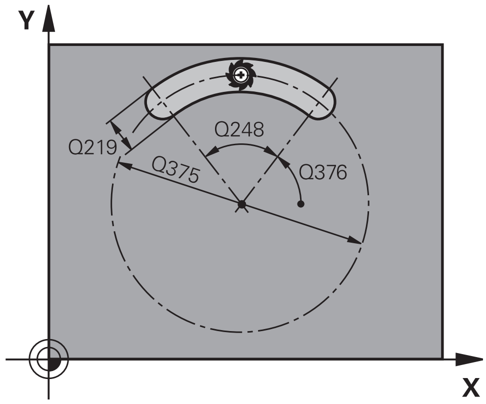
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

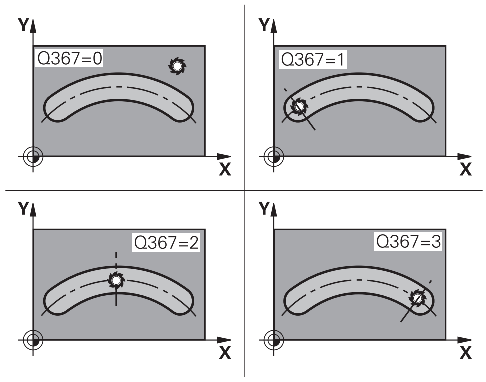
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

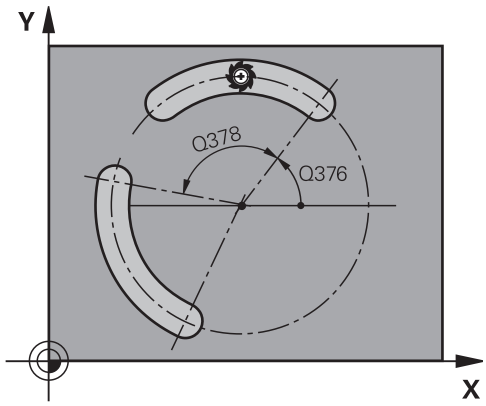
If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool to the first plunging depth at rapid traverse!

Cycle parameters

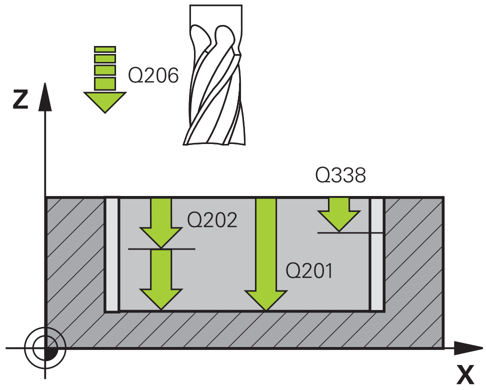
- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Slot width** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Pitch circle diameter** Q375: Enter the diameter of the pitch circle. Input range 0 to 99999.9999
- **Reference for slot position (0/1/2/3)** Q367: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle
  1: Tool position = center of left slot arc. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account
  2: Tool position = center of centerline. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account
  3: Tool position = center of right slot arc. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
- **Center in 1st axis** Q216 (absolute): Center of the pitch circle in the reference axis of the working plane. **Only effective if Q367 = 0**. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q217 (absolute): Center of the pitch circle in the minor axis of the working plane. **Only effective if Q367 = 0**. Input range -99999.9999 to 99999.9999
- **Starting angle** Q376 (absolute): Enter the polar angle of the starting point. Input range -360.000 to 360.000
- **Angular length** Q248 (incremental): Enter the angular length of the slot. Input range 0 to 360.000

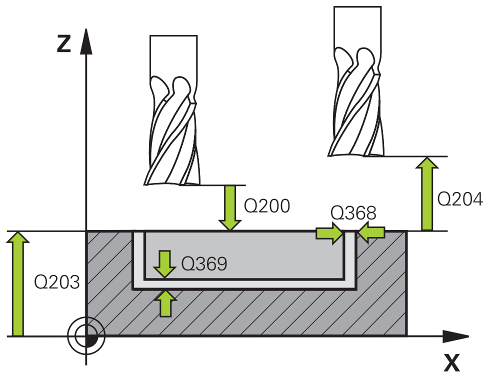






- **Stepping angle** Q378 (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. Input range -360.000 to 360.000
- **Number of repetitions** Q377: Number of machining operations on a pitch circle. Input range 1 to 99999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





NC blocks

8 CYCL DEF 254 CIRCULAR SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q375=80	;PITCH CIRCLE DIA.
Q367=0	;REF. SLOT POSITION
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q376=+45	;STARTING ANGLE
Q248=90	;ANGULAR LENGTH
Q378=0	;STEPPING ANGLE
Q377=1	;NR OF REPETITIONS
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR

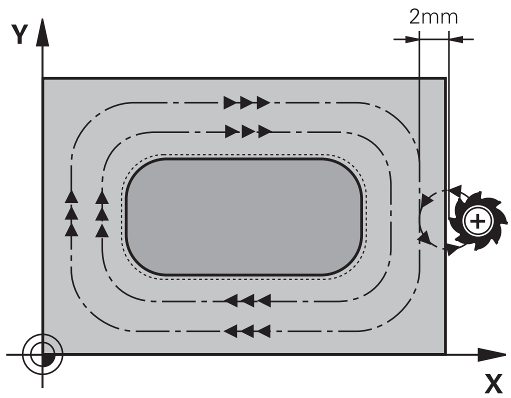
- **Plunging strategy** Q366: Type of plunging strategy:
  0: vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
  1, 2: reciprocal plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message
  PREDEF: The TNC uses the value from the GLOBAL DEF block
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate reference (0...3)** Q439: Define a reference for the programmed feed rate:
  0: The feed rate refers to the center point path of the tool
  1: The feed rate refers to the tool cutting edge only during side finishing; otherwise, it refers to the center point path
  2: The feed rate refers to the tool cutting edge during side **and** floor finishing; otherwise, it refers to the center point path
  3: The feed rate always refers to the tool cutting edge

```
Q206=150  ;FEED RATE FOR PLNGNG
Q338=5    ;INFEED FOR FINISHING
Q200=2    ;SET-UP CLEARANCE
Q203=+0   ;SURFACE COORDINATE
Q204=50   ;2ND SET-UP CLEARANCE
Q366=1    ;PLUNGE
Q385=500  ;FINISHING FEED RATE
Q439=0    ;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99
```

5.6 RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)

Cycle run

Use Cycle 256 to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the TNC performs multiple stepovers until the finished dimension has been machined.



1 The tool moves from the cycle starting position (stud center) to the starting position for stud machining. Specify the starting position with parameter Q437. The standard starting position (**Q437=0**) is 2 mm to the right of the unmachined stud
2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to the set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
3 The tool then moves tangentially to the stud contour and machines one revolution.
4 If the finished dimension cannot be machined with one revolution, the TNC performs a stepover with the current factor, and machines another revolution. The TNC takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached. If you have set the starting point on a corner (Q437 not equal to 0), the TNC mills on a spiral path from the starting point inward until the finished dimension has been reached.
5 If further stepovers are required, the tool then departs the contour on a tangential path and returns to the starting point of stud machining.
6 The TNC then plunges the tool to the next plunging depth, and machines the stud at this depth.
7 This process is repeated until the programmed stud depth is reached.
8 At the end of the cycle, the TNC merely positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

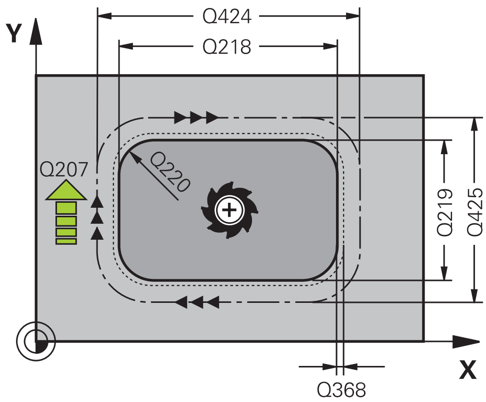
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

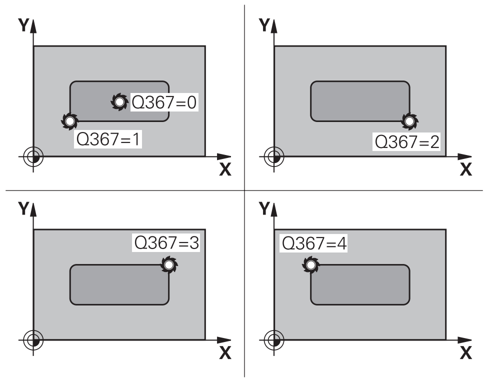
Depending on the approach position Q439, leave enough room next to the stud for the approach motion. At least tool diameter + 2 mm.

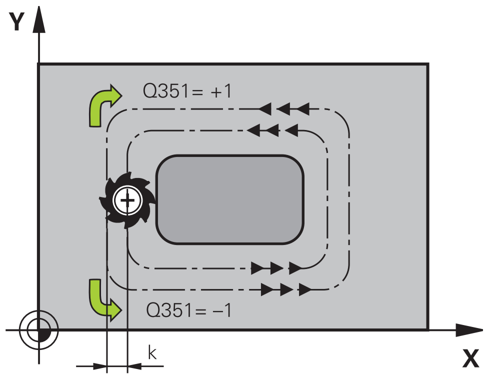
At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. This means that the end position of the tool after the cycle differs from the starting position.

Cycle parameters

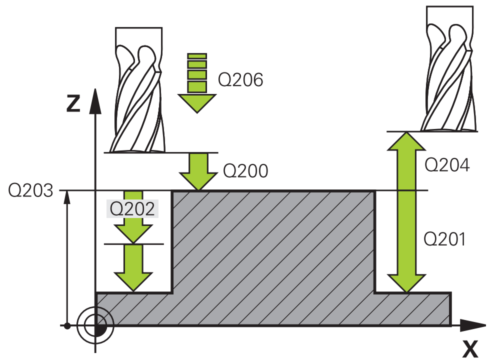
- **1st side length** Q218: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **Workpiece blank side length 1** Q424: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **1st side length.** The TNC performs multiple stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **2nd side length** Q219: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **2nd side length**. The TNC performs multiple stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **Workpiece blank side length 2** Q425: Length of the stud blank, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Corner radius** Q220: Radius of the stud corner. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane, is left over after machining. Input range 0 to 99999.9999
- **Angle of rotation** Q224 (absolute): Angle by which the entire machining is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.0000 to 360.0000
- **Stud position** Q367: Position of the stud in reference to the position of the tool when the cycle is called:
  0: Tool position = stud center
  1: Tool position = left corner below
  2: Tool position = right corner below
  3: Tool position = right corner top
  4: Tool position = left top corner top
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**







- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.9999; alternatively **PREDEF**
- **Approach position (0...4)** Q437: Define the approach strategy of the tool:
  0: Right of the stud (default setting)
  1: left corner below
  2: right corner below
  3: right corner top
  4: left corner top. If approach marks on the stud surface are caused by the setting Q437=0, specify another approach position.



NC blocks

8 CYCL DEF 256 RECTANGULAR STUD	
Q218=60	;FIRST SIDE LENGTH
Q424=74	;WORKPC. BLANK SIDE 1
Q219=40	;2ND SIDE LENGTH
Q425=60	;WORKPC. BLANK SIDE 2
Q220=5	;CORNER RADIUS
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;STUD POSITION
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q370=1	;TOOL PATH OVERLAP
Q437=0	;APPROACH POSITION
9 L X+50 Y+50 R0 FMAX M3 M99	

5.7 CIRCULAR STUD (Cycle 257, DIN/ISO: G257)

Cycle run

Use Cycle 257 to machine a circular stud. The TNC mills the circular stud with a helical infeed motion starting from the workpiece blank diameter.

1. If the tool is below the 2nd set-up clearance, the TNC retracts the tool to the 2nd set-up clearance.
2. The tool moves from the stud center to the starting position for stud machining. With the polar angle you specify the starting position with respect to the stud center using parameter Q376.
3. The TNC moves the tool at rapid traverse **FMAX** to the set-up clearance Q200, and from there advances to the first plunging depth at the feed rate for plunging.
4. The TNC then machines the circular stud with a helical infeed motion, taking the overlap factor into account.
5. The TNC retracts the tool from the contour by 2 mm on a tangential path.
6. If more than one plunging movement is required, the tool repeats the plunging movement at the point next to the departure movement.
7. This process is repeated until the programmed stud depth is reached.
8. At the end of the cycle, the tool departs on a tangential path and then retracts in the tool axis to the 2nd set-up clearance defined in the cycle.

Please note while programming:

Pre-position the tool in the machining plane to the starting position (stud center) with radius compensation **R0**.

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC returns the tool to the starting position.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

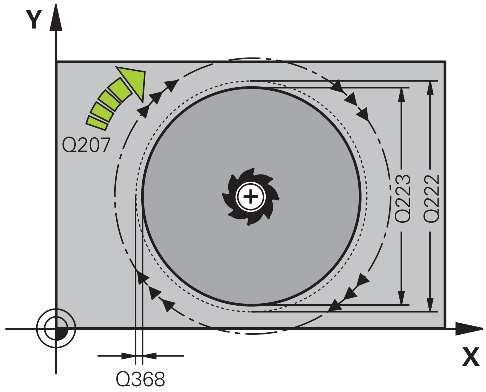
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

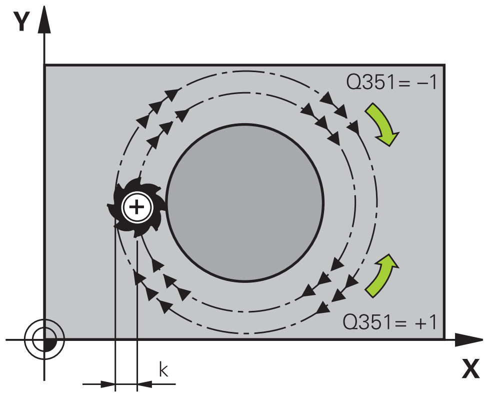
Depending on the starting angle Q376, leave enough room next to the stud for the approach motion. At least tool diameter + 2 mm.

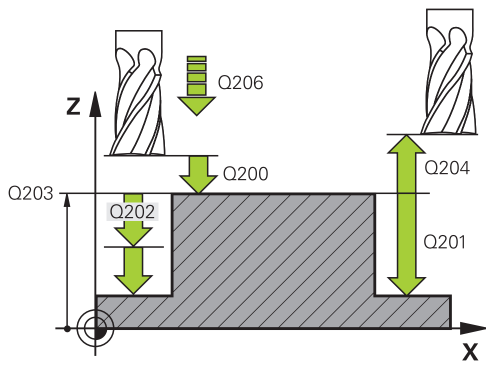
At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. This means that the end position of the tool after the cycle differs from the starting position.

Cycle parameters

- **Finished part diameter** Q223: Diameter of the completely machined stud. Input range 0 to 99999.9999
- **Workpiece blank diameter** Q222: Diameter of the workpiece blank. Enter the workpiece blank diameter greater than the finished diameter. The TNC performs multiple stepovers if the difference between the workpiece blank diameter and finished diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**







- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414 alternatively **PREDEF**
- **Starting angle** Q376: Polar angle relative to the stud center from which the tool approaches the stud. Input range 0 to 359°

NC blocks

```
8 CYCL DEF 257 CIRCULAR STUD
    Q223=60    ;FINISHED PART DIAMETER
    Q222=60    ;WORKPIECE BLANK DIAMETER
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q370=1     ;TOOL PATH OVERLAP
    Q376=0     ;STARTING ANGLE
9 L X+50 Y+50 R0 FMAX M3 M99
```

5.8 FACE MILLING (Cycle 233, DIN/ISO: G233)

Cycle run

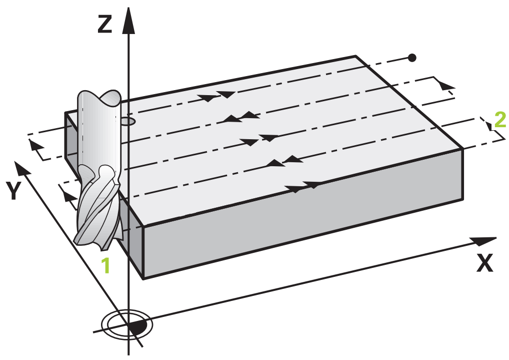
Cycle 233 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

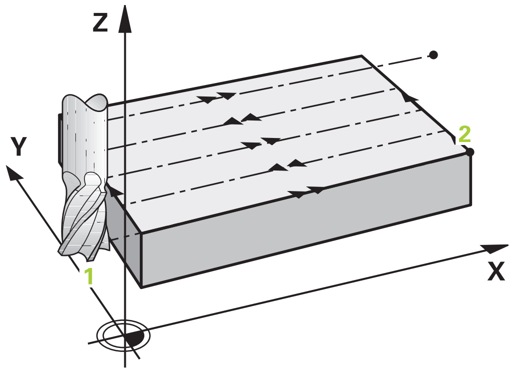
- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover when retracting at rapid traverse
- **Strategy Q389=3**: The surface is machined line by line without overtravel; stepover when retracting at rapid traverse
- **Strategy Q389=4**: Helical machining from the outside toward the inside

1 From the current position, the TNC positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane: The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2 The TNC then positions the tool at rapid traverse **FMAX** to the set-up clearance in the spindle axis.
3 The tool then moves in the tool axis at the feed rate for milling Q207 to the first plunging depth calculated by the TNC.

Strategies Q389=0 and Q389 =1

The strategies Q389=0 and Q389=1 differ in the overtravel during face milling. If Q389=0, the end point lies outside of the surface. If Q389=1, it lies at the edge of the surface. The TNC calculates the end point **2** from the side length and the safety clearance to the side. If the strategy Q389=0 is used, the TNC additionally moves the tool beyond the level surface by the tool radius.

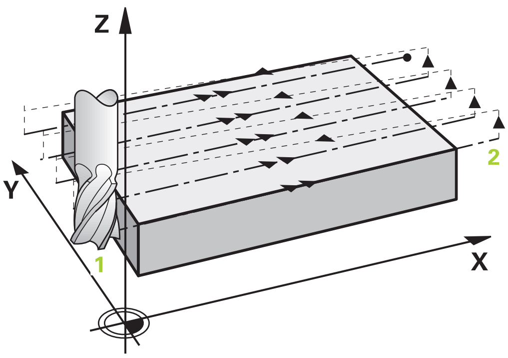


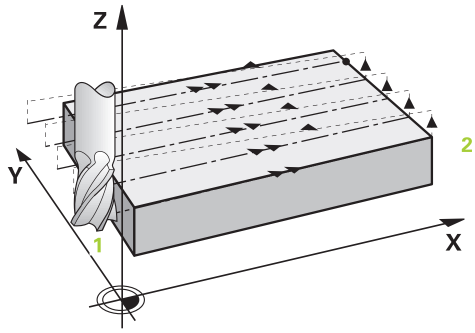


4 The TNC moves the tool to the end point **2** at the programmed feed rate for milling.
5 Then the TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and the safety clearance to the side.
6 The tool then returns at the feed rate for milling in the opposite direction.
7 The process is repeated until the programmed surface has been completed.
8 The TNC then positions the tool at rapid traverse **FMAX** back to the starting point **1**.
9 If more than one infeed is required, the TNC moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
10 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.
11 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategies Q389=2 and Q389 =3

The strategies Q389=2 and Q389=3 differ in the overtravel during face milling. If Q389=2, the end point lies outside of the surface. If Q389=3, it lies at the edge of the surface. The TNC calculates the end point **2** from the side length and the safety clearance to the side. If the strategy Q389=2 is used, the TNC additionally moves the tool beyond the level surface by the tool radius.

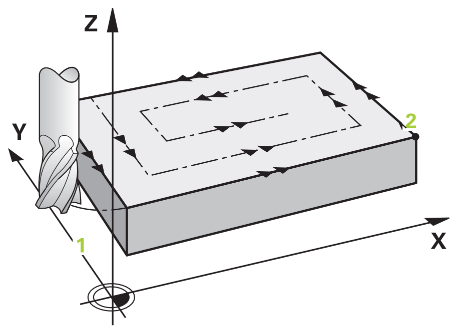




4 The tool subsequently advances to the end point **2** at the programmed feed rate for milling.
5 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at **FMAX** directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the safety clearance to the side.
6 The tool then returns to the current infeed depth and moves in the direction of the next end point **2**.
7 The multipass process is repeated until the programmed surface has been completed. At the end of the last path, the TNC positions the tool at rapid traverse **FMAX** back to the starting point **1**.
8 If more than one infeed is required, the TNC moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
9 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.
10 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

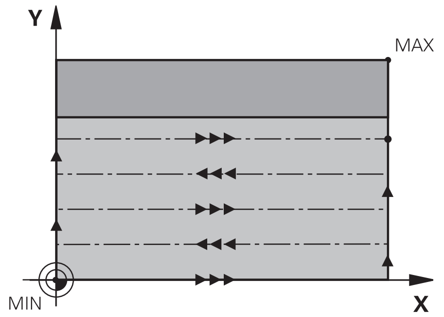
Strategy Q389=4

4 The tool subsequently approaches the starting point of the milling path on a tangential arc at the programmed **feed rate for milling**.
5 The TNC machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged.
6 The process is repeated until the programmed surface has been completed. At the end of the last path, the TNC positions the tool at rapid traverse **FMAX** back to the starting point **1**.
7 If more than one infeed is required, the TNC moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
8 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.



Limits

The limits enable you to set limits to the machining of the level surface so that, for example, side walls or shoulders are considered during machining. A side wall that is defined by a limit is machined to the finished dimension resulting from the starting point or the side lengths of the level surface. During roughing the TNC takes the allowance for the side into account, whereas during finishing the allowance is used for pre-positioning the tool.



Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Keep in mind the machining direction.

The TNC automatically pre-positions the tool in the tool axis. Note the **2nd** set-up clearance Q204.

Enter the **2nd set-up clearance** in Q204 so that no collision with the workpiece or the fixtures can occur.

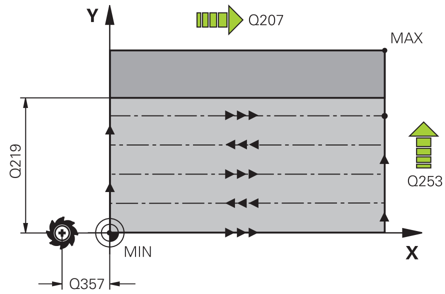
If the starting point in the 3rd axis Q227 and the end point in the 3rd axis Q386 are entered as equal values, the TNC does not run the cycle (depth = 0 has been programmed).

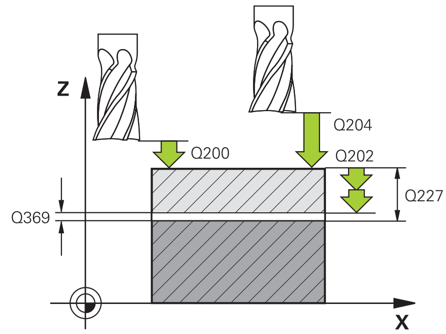
Danger of collision!

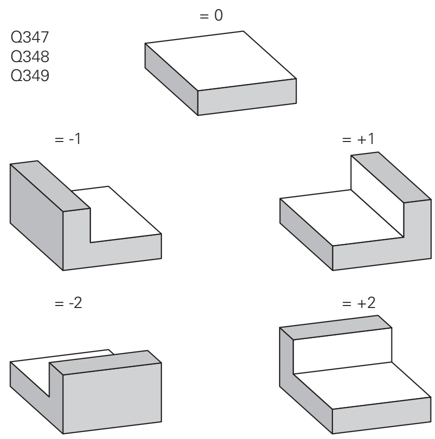
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning if starting point < end point is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!

Cycle parameters







- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Milling strategy (0 to 4)** Q389: Determine how the TNC should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface to be machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate outside the surface to be machined
  3: Line-by-line machining, retraction and stepover at the positioning feed rate at the edge of the surface to be machined
  4: Helical machining, uniform infeed from the outside toward the inside
- **Milling direction** Q350: Axis in the machining plane that defines the machining direction:
  1: Reference axis = machining direction
  2: Minor axis = machining direction
- **1st side length** Q218 (incremental): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in the 1st axis. Input range 0 to 99999.9999
- **2nd side length** Q219 (incremental value): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **starting point in the 2nd axis**. Input range -99999.9999 to 99999.9999

- **Starting point in 3rd axis** Q227 (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999
- **End point in 3rd axis** Q386 (absolute): Coordinate in the spindle axis to which the surface is to be face milled. Input range -99999.9999 to 99999.9999
- **Allowance for floor** Q369 (incremental): Distance used for the last infeed. Input range 0 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Path overlap factor** Q370: Maximum stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. Input range: 0.1 bis 1.9999.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min, while milling the last infeed. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207. Input range 0 to 99999.9999, alternatively **FMAX**, **FAUTO**
- **Clearance to side** Q357 (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**

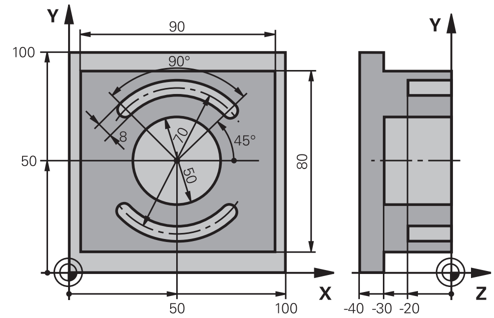
NC blocks

8 CYCL DEF 233 FACE MILLING	
Q215=0	;MACHINING OPERATION
Q389=2	;MILLING STRATEGY
Q350=1	;MILLING DIRECTION
Q218=120	;1ST SIDE LENGTH
Q219=80	;2ND SIDE LENGTH
Q227=0	;STARTNG PNT 3RD AXIS
Q386=-6	;END POINT 3RD AXIS
Q369=0.2	;ALLOWANCE FOR FLOOR
Q202=3	;MAX. PLUNGING DEPTH
Q370=1	;TOOL PATH OVERLAP
Q207=500	;FEED RATE FOR MILLING
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO THE SIDE
Q200=2	;SET-UP CLEARANCE
Q204=50	;2ND SET-UP CLEARANCE
Q347=0	;1ST LIMIT
Q348=0	;2ND LIMIT
Q349=0	;3RD LIMIT
Q220=0	;CORNER RADIUS
Q368=0	;ALLOWANCE FOR SIDE
Q338=0	;INFEED FOR FINISHING
9 L X+0 Y+0 R0 FMAX M3 M99	

- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **1st limit** Q347:Select the workpiece side on which the level surface is limited by a side wall (not possible for helical machining). Depending on the position of the side wall, the TNC limits the machining of the level surface to the respective coordinate of the starting point or to the side length: (not possible for helical machining):
  Input **0**: No limit
  Input **-1**: Limit in the negative reference axis
  Input **+1**: Limit in the positive reference axis
  Input **-2**: Limit in the negative minor axis
  Input **+2**: Limit in the positive minor axis
- **2nd limit** Q348: See parameter 1st limit Q347
- **3rd limit** Q349: See parameter 1st limit Q347
- **Corner radius** Q220: Radius for corner at limits (Q347 to Q349). Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999

5.9 Programming Examples

Example: Milling pockets, studs and slots



0 BEGINN PGM C210 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 1 Z S3500	Call the tool for roughing/finishing
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Define cycle for machining the contour outside
Q218=90 ;FIRST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q220=0 ;CORNER RADIUS	
Q368=0 ;ALLOWANCE FOR SIDE	
Q224=0 ;ANGLE OF ROTATION	
Q367=0 ;STUD POSITION	
Q207=250 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q437=0 ;APPROACH POSITION	
6 L X+50 Y+50 R0 M3 M99	Call cycle for machining the contour outside
7 CYCL DEF 252 CIRCULAR POCKET	Define CIRCULAR POCKET MILLING cycle
Q215=0 ;MACHINING OPERATION	
Q223=50 ;CIRCLE DIAMETER	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q207=500 ;FEED RATE FOR MILLING	

Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q366=1 ;PLUNGE	
Q385=750 ;FINISHING FEED RATE	
8 L X+50 Y+50 R0 FMAX M99	Call CIRCULAR POCKET MILLING cycle
9 L Z+250 R0 FMAX M6	Tool change
10 TOLL CALL 2 Z S5000	Call tool: slotting mill
11 CYCL DEF 254 CIRCULAR SLOT	Define SLOT cycle
Q215=0 ;MACHINING OPERATION	
Q219=8 ;SLOT WIDTH	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q375=70 ;PITCH CIRCLE DIA.	
Q367=0 ;REF. SLOT POSITION	No pre-positioning in X/Y required
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q376=+45 ;STARTING ANGLE	
Q248=90 ;ANGULAR LENGTH	
Q378=180 ;STEPPING ANGLE	Starting point for second slot
Q377=2 ;NR OF REPETITIONS	
Q207=500 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-20 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGE	
12 CYCL CALL FMAX M3	Call SLOT cycle
13 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
14 END PGM C210 MM	

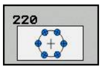
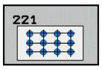
6

Fixed Cycles: Pattern Definitions

6.1 Fundamentals

Overview

The TNC provides two cycles for machining point patterns directly:

Cycle	Soft key	Page
220 POLAR PATTERN		179
221 CARTESIAN PATTERN		182

You can combine Cycle 220 and Cycle 221 with the following fixed cycles:

> If you have to machine irregular point patterns, use **CYCL CALL PAT** (see "Point tables", page 69) to develop point tables.
>
> More regular point patterns are available with the **PATTERN DEF** function (see "PATTERN DEF pattern definition", page 62).

Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL DRILLING
Cycle 204	BACK BORING
Cycle 205	UNIVERSAL PECKING
Cycle 206	TAPPING NEW with a floating tap holder
Cycle 207	RIGID TAPPING without a floating tap holder NEW
Cycle 208	BORE MILLING
Cycle 209	TAPPING WITH CHIP BREAKING
Cycle 240	CENTERING
Cycle 251	RECTANGULAR POCKET
Cycle 252	CIRCULAR POCKET MILLING
Cycle 253	SLOT MILLING
Cycle 254	CIRCULAR SLOT (can only be combined with Cycle 221)
Cycle 256	RECTANGULAR STUD
Cycle 257	CIRCULAR STUD
Cycle 262	THREAD MILLING
Cycle 263	THREAD MILLING/COUNTERSINKING
Cycle 264	THREAD DRILLING/MILLING
Cycle 265	HELICAL THREAD DRILLING/MILLING
Cycle 267	OUTSIDE THREAD MILLING

6.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220)

Cycle run

1 At rapid traverse, the TNC moves the tool from its current position to the starting point for the first machining operation.
  Sequence:
  - 2. Move to the 2nd set-up clearance (spindle axis)
  - Approach the starting point in the spindle axis.
  - Move to the set-up clearance above the workpiece surface (spindle axis).
2 From this position, the TNC executes the last defined fixed cycle.
3 The tool then approaches on a straight line or circular arc the starting point for the next machining operation. The tool stops at the set-up clearance (or the 2nd set-up clearance).
4 This process (1 to 3) is repeated until all machining operations have been executed.

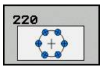
Please note while programming:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

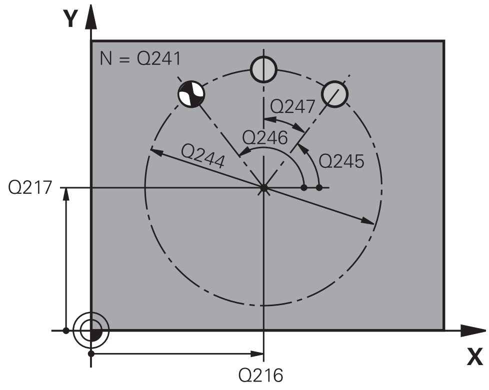
If you combine Cycle 220 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 220 will be effective for the selected fixed cycle.

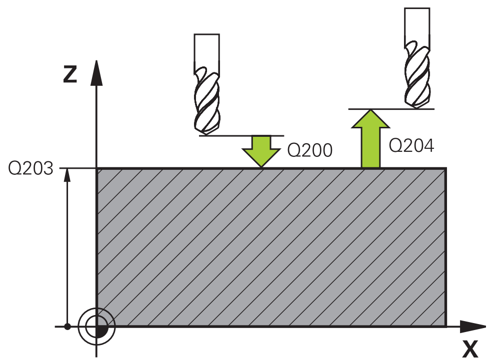
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Center in 1st axis** Q216 (absolute): Center of the pitch circle in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q217 (absolute): Center of the pitch circle in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Pitch circle diameter** Q244: Diameter of the pitch circle. Input range 0 to 99999.9999
- **Starting angle** Q245 (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range -360.000 to 360.000
- **Stopping angle** Q246 (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to full circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). Input range -360.000 to 360.000
- **Number of repetitions** Q241: Number of machining operations on a pitch circle. Input range 1 to 99999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999





NC blocks

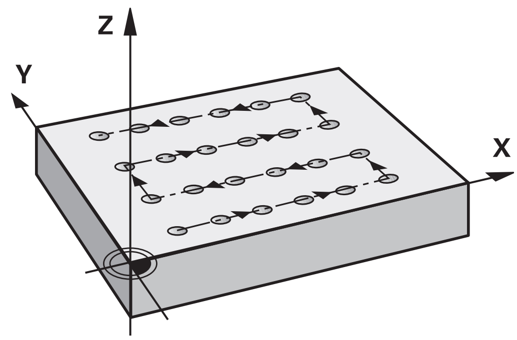
```
53 CYCL DEF 220 POLAR PATTERN
    Q216=+50   ;CENTER IN 1ST AXIS
    Q217=+50   ;CENTER IN 2ND AXIS
    Q244=80    ;PITCH CIRCLE DIA.
    Q245=+0    ;STARTING ANGLE
    Q246=+360;STOPPING ANGLE
    Q247=+0    ;STEPPING ANGLE
    Q241=8     ;NR OF REPETITIONS
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
```

- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between machining operations:
  0: Move at set-up clearance between machining operations
  1: Move at 2nd set-up clearance between machining operations
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool moves between machining operations:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE
Q365=0	;TYPE OF TRAVERSE

6.3 LINEAR PATTERN (Cycle 221, DIN/ISO: G221)

Cycle run



1. The TNC automatically moves the tool from its current position to the starting point for the first machining operation.
   Sequence:
   - 2. Move to the set-up clearance (spindle axis)
   - Approach the starting point in the machining plane
   - Move to the set-up clearance above the workpiece surface (spindle axis)
2. From this position, the TNC executes the last defined fixed cycle.
3. The tool then approaches the starting point for the next machining operation in the positive reference axis direction at set-up clearance (or 2nd set-up clearance).
4. This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line.
5. The tool subsequently moves to the last point on the second line where it carries out the machining operation.
6. From this position, the tool approaches the starting point for the next machining operation in the negative reference axis direction.
7. This process (6) is repeated until all machining operations in the second line have been executed.
8. The tool then moves to the starting point of the next line.
9. All subsequent lines are processed in a reciprocating movement.

Please note while programming:

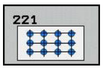
Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.

If you combine Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface, 2nd set-up clearance and the rotational position that you defined in Cycle 221 will be effective for the selected fixed cycle.

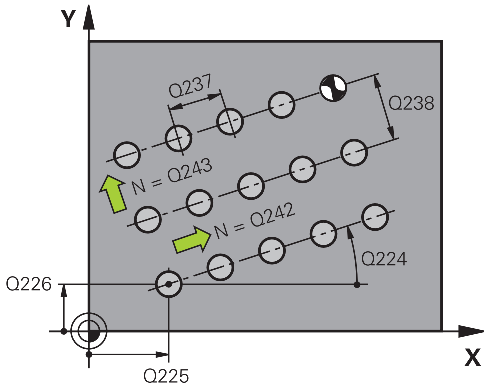
The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

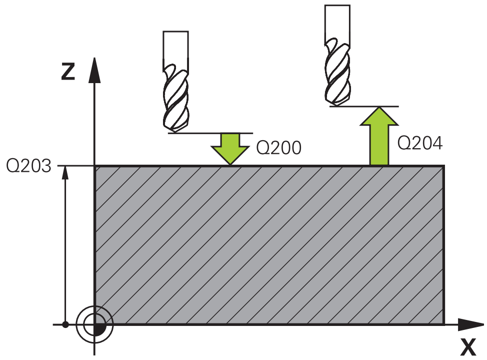
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Starting point in 1st axis** Q225 (absolute): Coordinate of the starting point in the reference axis of the working plane.
- **Starting point 2nd axis** Q226 (absolute): Coordinate of the starting point in the minor axis of the machining plane
- **Spacing in 1st axis** Q237 (incremental): Spacing between each point on a line
- **Spacing in 2nd axis** Q238 (incremental): Spacing between each line
- **Number of columns** Q242: Number of machining operations on a line
- **Number of lines** Q243: Number of lines
- **Angle of rotation** Q224 (absolute): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between machining operations:
  0: Move at set-up clearance between machining operations
  1: Move at 2nd set-up clearance between machining operations



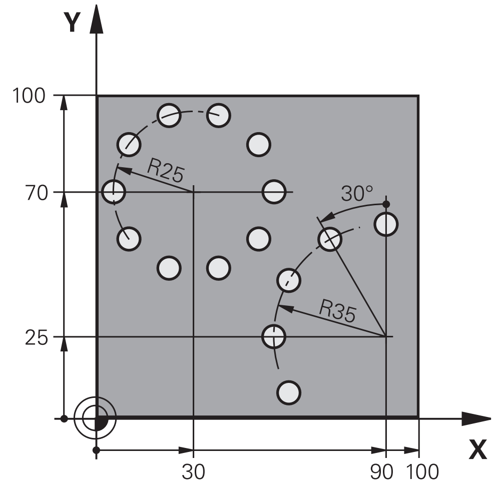


NC blocks

54 CYCL DEF 221 CARTESIAN PATTERN	
Q225=+15	;STARTNG PNT 1ST AXIS
Q226=+15	;STARTNG PNT 2ND AXIS
Q237=+10	;SPACING IN 1ST AXIS
Q238=+8	;SPACING IN 2ND AXIS
Q242=6	;NUMBER OF COLUMNS
Q243=4	;NUMBER OF LINES
Q224=+15	;ANGLE OF ROTATION
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

6.4 Programming Examples

Example: Polar hole patterns



0 BEGIN PGM BOHRB MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 Y+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call
4 L Z+250 R0 FMAX M3	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT BOTTOM	
6 CYCL DEF 220 POLAR PATTERN	Define cycle for polar pattern 1, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q216=+30 ;CENTER IN 1ST AXIS	
Q217=+70 ;CENTER IN 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIA.	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	

Q301=1	;MOVE TO CLEARANCE	
Q365=0	;TYPE OF TRAVERSE	
7 CYCL DEF 220 POLAR PATTERN		Define cycle for polar pattern 2, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q216=+90	;CENTER IN 1ST AXIS	
Q217=+25	;CENTER IN 2ND AXIS	
Q244=70	;PITCH CIRCLE DIA.	
Q245=+90	;STARTING ANGLE	
Q246=+360	;STOPPING ANGLE	
Q247=+30	;STEPPING ANGLE	
Q241=5	;NR OF REPETITIONS	
Q200=2	;SET-UP CLEARANCE	
Q203=+0	;SURFACE COORDINATE	
Q204=100	;2ND SET-UP CLEARANCE	
Q301=1	;MOVE TO CLEARANCE	
Q365=0	;TYPE OF TRAVERSE	
8 L Z+250 R0 FMAX M2		Retract in the tool axis, end program
9 END PGM BOHRB MM		

7

Fixed Cycles: Contour Pocket

7.1 SL Cycles

Fundamentals

SL cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle 14 CONTOUR GEOMETRY.

> The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
>
> SL cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always run a graphical program test before machining! This is a simple way of finding out whether the TNC-calculated program will provide the desired results.
>
> When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL.
- The subprograms must not contain spindle axis coordinates.
- Always program both axes in the first block of the subprogram
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms.

Program structure: Machining with SL cycles

```
0 BEGIN PGM SL2 MM
...
12 CYCL DEF 14 CONTOUR...
13 CYCL DEF 20 CONTOUR DATA...
...
16 CYCL DEF 21 PILOT DRILLING...
17 CYCL CALL
...
18 CYCL DEF 22 ROUGH-OUT...
19 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
```

```
...
99 END PGM SL2 MM
```

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before each cycle–position the tool to a safe position before the cycle call.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies to the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Overview

Cycle	Soft key	Page
14 CONTOUR GEOMETRY (essential)	14 LBL 1...N	190
20 CONTOUR DATA (essential)	20 CONTOUR DATA	195
21 PILOT DRILLING (optional)	21	197
22 ROUGH-OUT (essential)	22	199
23 FLOOR FINISHING (optional)	23	203
24 SIDE FINISHING (optional)	24	205

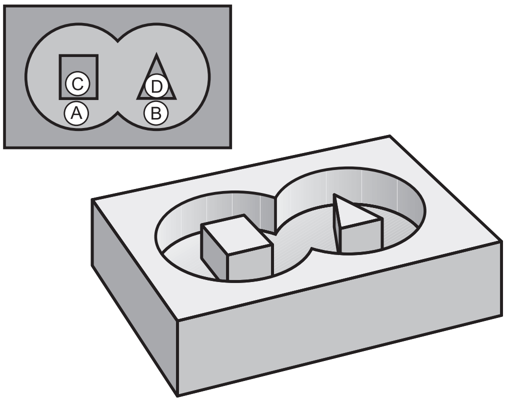
Enhanced cycles:

Cycle	Soft key	Page
25 CONTOUR TRAIN	25	208
270 CONTOUR TRAIN DATA	270	210

7.2 CONTOUR (Cycle 14, DIN/ISO: G37)

Please note while programming:

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY.



> Cycle 14 is DEF active which means that it becomes effective as soon as it is defined in the part program.
>
> You can list up to 12 subprograms (subcontours) in Cycle 14.

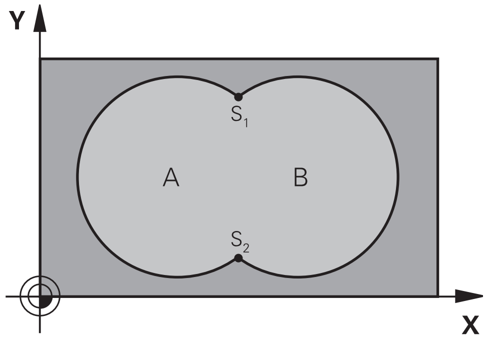
Cycle parameters

- **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key. Entry of up to 12 subprogram numbers 1 to 65535.

7.3 Superimposed contours

Fundamentals

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



NC blocks

```
12 CYCL DEF 14.0 CONTOUR
13 CYCL DEF 14.1 CONTOUR LABEL
   1/2/3/4
```

Subprograms: overlapping pockets

> The subsequent programming examples are contour subprograms that are called by Cycle 14 CONTOUR GEOMETRY in a main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Pocket A

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

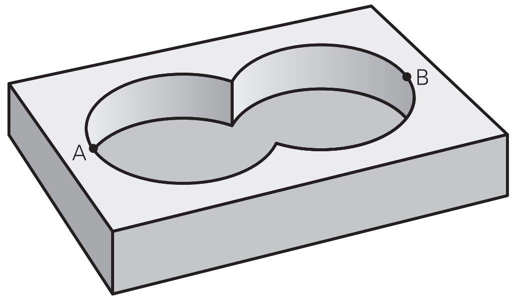
Subprogram 2: Pocket B

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

Area of inclusion

Both surfaces A and B are to be machined, including the overlapping area:

- The surfaces A and B must be pockets.
- The first pocket (in Cycle 14) must start outside the second pocket.



Surface A:

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

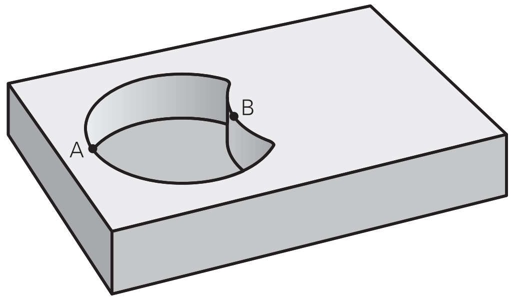
Surface B:

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

Area of exclusion

Surface A is to be machined without the portion overlapped by B:

- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.



Surface A:

51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0

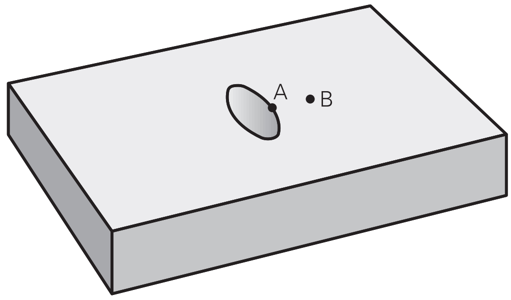
Surface B:

56 LBL 2
57 L X+40 Y+50 RL
58 CC X+65 Y+50
59 C X+40 Y+50 DR-
60 LBL 0

Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- A and B must be pockets.
- A must start inside of B.



Surface A:

```
51 LBL 1
52 L X+60 Y+50 RR
53 CC X+35 Y+50
54 C X+60 Y+50 DR-
55 LBL 0
```

Surface B:

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

7.4 CONTOUR DATA (Cycle 20, DIN/ISO: G120)

Please note while programming:

Machining data for the subprograms describing the subcontours are entered in Cycle 20.

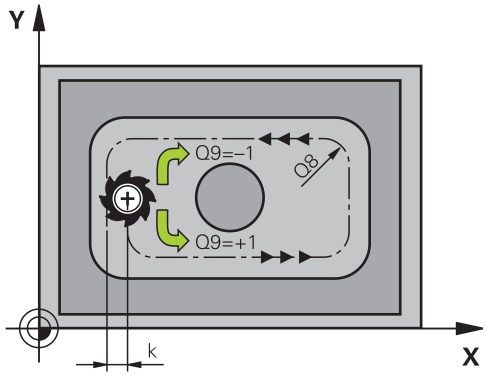
> Cycle 20 is DEF active, which means that it becomes effective as soon as it is defined in the part program.
>
> The machining data entered in Cycle 20 are valid for Cycles 21 to 24.
>
> The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the TNC performs the cycle at the depth 0.
>
> If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q20 cannot be used as program parameters.

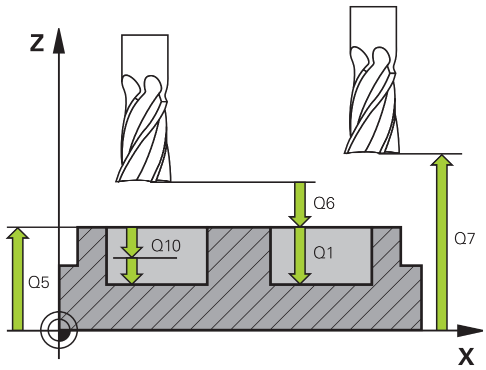
Cycle parameters

20 CONTOUR DATA

- **Milling depth** Q1 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Path overlap** factor Q2: Q2 x tool radius = stepover factor k. Input range -0.0001 to 1.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the working plane. Input range -99999.9999 to 99999.9999
- **Finishing allowance for floor** Q4 (incremental): Finishing allowance in the tool axis. Input range -99999.9999 to 99999.9999
- **Workpiece surface coordinate** Q5 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Clearance height** Q7 (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range -99999.9999 to 99999.9999
- **Inside corner radius** Q8: Inside "corner" rounding radius; entered value is referenced to the path of the tool center and is used to calculate smoother traverse motions between the contour elements. **Q8 is not a radius that is inserted as a separate contour element between programmed elements!** Input range 0 to 99999.9999
- **Direction of rotation?** Q9: Machining direction for pockets
  - Q9 = –1 up-cut milling for pocket and island
  - Q9 = +1 climb milling for pocket and island

You can check the machining parameters during a program interruption and overwrite them if required.





NC blocks

57 CYCL DEF 20 CONTOUR DATA	
Q1=-20	;MILLING DEPTH
Q2=1	;TOOL PATH OVERLAP
Q3=+0.2	;ALLOWANCE FOR SIDE
Q4=+0.1	;ALLOWANCE FOR FLOOR
Q5=+30	;SURFACE COORDINATE
Q6=2	;SET-UP CLEARANCE
Q7=+80	;CLEARANCE HEIGHT
Q8=0.5	;ROUNDING RADIUS
Q9=+1	;DIRECTION OF ROTATION

7.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121)

Cycle run

Use Cycle 21 PILOT DRILLING if you will subsequently rough out the contour with a tool other than a center-cut end mill (ISO 1641). This cycle drills a hole in the area that is to be roughed out with a cycle such as Cycle 22. Cycle 21 takes the allowance for side and the allowance for floor as well as the radius of the rough-out tool into account for the cutter infeed points. The cutter infeed points also serve as starting points for roughing.

Before calling Cycle 21 you need to program two further cycles:

- **Cycle 14 CONTOUR GEOMETRY** or SEL CONTOUR—needed by Cycle 21 PILOT DRILLING in order to determine the drilling position in the plane
- **Cycle 20 CONTOUR DATA**—needed by Cycle 21 PILOT DRILLING in order to determine parameters such as hole depth and set-up clearance

Cycle run:

1. The TNC first positions the tool in the plane (the position results from the contour you have defined with Cycle 14 or SEL CONTOUR, and from the rough-out tool data).
2. The tool then moves at rapid traverse **FMAX** to the set-up clearance. (Define the set-up clearance in Cycle 20 CONTOUR DATA).
3. The tool drills from the current position to the first plunging depth at the programmed feed rate **F**.
4. Then the tool retracts at rapid traverse **FMAX** to the starting position and advances again to the first plunging depth minus the advanced stop distance t.
5. The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: t = 0.6 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
6. The tool then advances with another infeed at the programmed feed rate **F**.
7. The TNC repeats this process (1 to 4) until the programmed total hole depth is reached. The finishing allowance for floor is taken into account.
8. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.

Please note while programming:

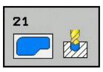
When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **TOOL CALL** block.

In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

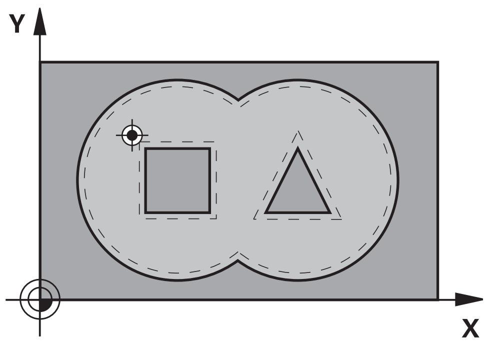
If Q13=0, the TNC uses the data of the tool that is currently in the spindle.

At the end of the cycle, move the tool in the plane to an absolute position, not to an incremental position, if you have set the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket to ToolAxClearanceHeight.

Cycle parameters



- **Plunging depth** Q10 (incremental): Dimension by which the tool drills in each infeed (negative sign for negative working direction). Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool when plunging into the workpiece in mm/min. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**, **FZ**
- **Rough-out tool number/name** Q13 or QS13: Number or name of rough-out tool. Input range 0 to 32767.9 if a number is entered; maximum 16 characters if a name is entered. If you enter Q13=0, the TNC uses the data of the tool that is currently in the spindle.



NC blocks

58 CYCL DEF 21 PILOT DRILLING	
Q10=+5	**;PLUNGING DEPTH**
Q11=100	**;FEED RATE FOR PLNGNG**
Q13=1	**;ROUGH-OUT TOOL**

7.6 ROUGHING (Cycle 22, DIN/ISO: G122)

Cycle run

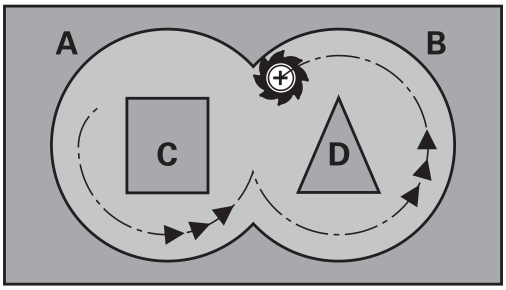
Use Cycle 22 ROUGHING to define the technology data for roughing.

Before calling Cycle 22 you need to program further cycles:

- Cycle 14 CONTOUR GEOMETRY or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary

Cycle run

1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2. In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate.
3. First the island contours (C and D in the figure at right) are rough-milled until the pocket contour (A, B) is approached.
4. In the next step the TNC moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached.
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.



Please note while programming:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

You define the plunging behavior of Cycle 22 with parameter Q19 and with the tool table in the **ANGLE** and **LCUTS** columns:

- If Q19=0 is defined, the TNC always plunges perpendicularly, even if a plunge angle (**ANGLE**) is defined for the active tool.
- If you define the **ANGLE**=90°, the TNC plunges perpendicularly. The reciprocation feed rate Q19 is used as plunging feed rate.
- If the reciprocation feed rate Q19 is defined in Cycle 22 and **ANGLE** is defined between 0.1 and 89.999 in the tool table, the TNC plunges helically at the defined **ANGLE**.
- If the reciprocation feed is defined in Cycle 22 and no **ANGLE** is in the tool table, the TNC displays an error message.
- If geometrical conditions do not allow helical plunging (slot), the TNC tries a reciprocating plunge. The reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**).

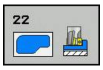
If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.

During fine roughing the TNC does not take a defined wear value **DR** of the coarse roughing tool into account.

Danger of collision!

After executing an SL cycle you must program the first traverse motion in the working plane with both coordinate data, e.g. **L X+80 Y +0 R0 FMAX**. At the end of the cycle, move the tool in the plane to an absolute position, not to an incremental position, if you have set the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket to ToolAxClearanceHeight.

Cycle parameters



- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Coarse roughing tool** Q18 or QS18: Number or name of the tool with which the TNC has already coarse-roughed the contour. Switch to the name input: Press the **TOOL NAME** soft key. The TNC automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the tool table TOOL.T and define the maximum plunging **ANGLE** of the tool. Otherwise, the TNC will display an error message. Input range 0 to 99999 if a number is entered; maximum 16 characters if a name is entered.
- **Reciprocation feed rate** Q19: Traversing speed of the tool in mm/min during reciprocating plunge cut. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting after machining. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q12. Input range 0 to 99999.9999, alternatively **FMAX,FAUTO**

NC blocks

59 CYCL DEF 22 ROUGH-OUT	
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=750	;FEED RATE FOR MILLING
Q18=1	;COARSE ROUGHING TOOL
Q19=150	;RECIPROCATION FEED RATE
Q208=9999	;RETRACTION FEED RATE
Q401=80	;FEED RATE REDUCTION
Q404=0	;FINE ROUGH STRATEGY

- **Feed rate factor in %** Q401: Percentage factor by which the TNC reduces the machining feed rate (Q12) as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (Q2) specified in Cycle 20. The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machining time should be reduced in total. Input range 0.0001 to 100.0000
- **Fine rough strategy** Q404: Specify the fine roughing behavior of the TNC if the radius of the fine roughing tool is greater than half the diameter of the rough-out tool:
  Q404=0:
  The TNC moves the tool between the areas to be fine roughed at the current depth along the contour
  Q404=1:
  The TNC retracts the tool to the set-up clearance between the areas to be fine roughed and then moves to the starting point for the next area to be roughed out.

7.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123)

Cycle run

With Cycle 23 FLOOR FINISHING, you can clear the finishing allowance for floor that is programmed in Cycle 20. The tool approaches the machining plane smoothly (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the TNC moves the tool to depth vertically. The tool then clears the finishing allowance remaining from rough-out.

Before calling Cycle 23 you need to program further cycles:

- Cycle 14 CONTOUR GEOMETRY or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary
- Cycle 22 ROUGHING, if necessary

Cycle run

1. The TNC positions the tool to the clearance height at rapid traverse FMAX.
2. The tool then moves in the tool axis at the feed rate Q11.
3. The tool approaches the machining plane smoothly (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the TNC moves the tool to depth vertically.
4. The tool clears the finishing allowance remaining from rough-out.
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.

Please note while programming:

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.

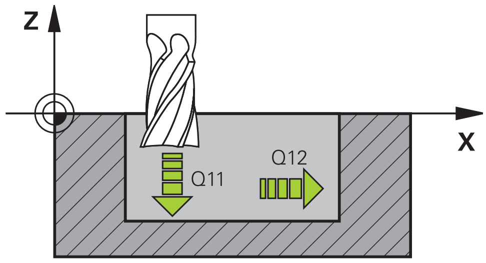
Danger of collision!

After executing an SL cycle you must program the first traverse motion in the working plane with both coordinate data, e.g. **L X+80 Y+0 R0 FMAX**.

At the end of the cycle, move the tool in the plane to an absolute position, not to an incremental position, if you have set the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket to ToolAxClearanceHeight.

Cycle parameters

- **Feed rate for plunging** Q11: Traversing speed of the tool when plunging into the workpiece in mm/min. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting after machining. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q12. Input range 0 to 99999.9999, alternatively **FMAX,FAUTO**



NC blocks

```
60 CYCL DEF 23 FLOOR FINISHING
    Q11=100   ;FEED RATE FOR PLNGNG
    Q12=350   ;FEED RATE FOR MILLING
    Q208=9999;RETRACTION FEED RATE
```

7.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124)

Cycle run

With Cycle 24 SIDE FINISHING, you can clear the finishing allowance for side that is programmed in Cycle 20. You can run this cycle in climb or up-cut milling.

Before calling Cycle 24 you need to program further cycles:

- Cycle 14 CONTOUR GEOMETRY or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary
- Cycle 22 ROUGHING, if necessary

Cycle run

1. The TNC positions the tool above the workpiece surface to the starting point for the approach position. This position in the plane results from a tangential arc on which the TNC moves the tool when approaching the contour.
2. The tool then advances to the first plunging depth at the feed rate for plunging.
3. The contour is approached on a tangential arc until the entire contour is completed. Each subcontour is finished separately.
4. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.

Please note while programming:

The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle 20) and the radius of the rough mill.

If no allowance has been defined in Cycle 20, the control issues the error message "Tool radius too large".

The allowance for side Q14 is left over after finishing. Therefore, it must be smaller than the allowance in Cycle 20.

This calculation also holds if you run Cycle 24 without having roughed out with Cycle 22; in this case, enter "0" for the radius of the rough mill.

You can use Cycle 24 also for contour milling. Then you must:

- define the contour to be milled as a single island (without pocket limit), and
- enter the finishing allowance (Q3) in Cycle 20 to be greater than the sum of the finishing allowance Q14 + radius of the tool being used.

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle 20.

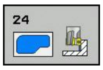
The starting point calculated by the TNC also depends on the machining sequence. If you select the finishing cycle with the GOTO key and then start the program, the starting point can be at a different location from where it would be if you execute the program in the defined sequence.

Danger of collision!

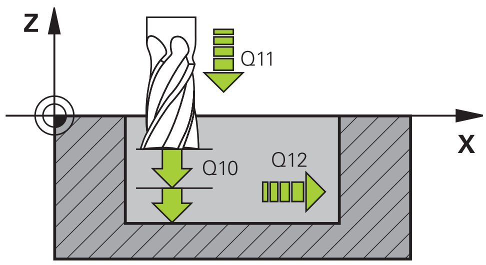
After executing an SL cycle you must program the first traverse motion in the working plane with both coordinate data, e.g. **L X+80 Y+0 R0 FMAX**.

At the end of the cycle, move the tool in the plane to an absolute position, not to an incremental position, if you have set the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket to ToolAxClearanceHeight.

Cycle parameters



- **Direction of rotation** Q9: Machining direction:
  +1: Rotation counterclockwise
  –1: Rotation clockwise
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool when plunging into the workpiece in mm/min. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Finishing allowance for side** Q14 (incremental): The allowance for side Q14 is left over after finishing. (This allowance must be smaller than the allowance in Cycle 20.) Input range -99999.9999 to 99999.9999

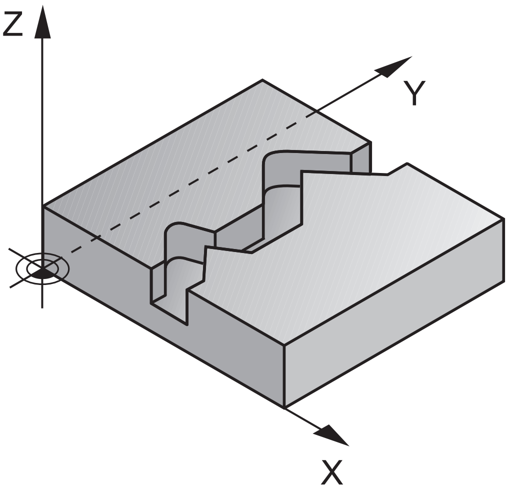


NC blocks

61 CYCL DEF 24 SIDE FINISHING	
Q9=+1	;DIRECTION OF ROTATION
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q14=+0	;ALLOWANCE FOR SIDE

7.9 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125)

Cycle run



In conjunction with Cycle 14 CONTOUR GEOMETRY, this cycle facilitates the machining of open and closed contours.

Cycle 25 CONTOUR TRAIN offers considerable advantages over machining a contour using positioning blocks:

- The TNC monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked.
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are mirrored.
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining.
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

Please note while programming:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The TNC takes only the first label of Cycle 14 CONTOUR GEOMETRY into account.

The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.

Cycle 20 **CONTOUR DATA** is not required.

The miscellaneous functions **M109** and **M110** are not effective when machining a contour with Cycle 25.

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

> **!**
>
> **Danger of collision!**
>
> To avoid collisions,
>
> - Do not program positions in incremental dimensions immediately after Cycle 25 since they are referenced to the position of the tool at the end of the cycle.
> - Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.

Cycle parameters

- **Milling depth** Q1 (incremental): Distance between workpiece surface and contour floor. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the working plane. Input range -99999.9999 to 99999.9999
- **Workpiece surface coordinate** Q5 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Clearance height** Q7 (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range -99999.9999 to 99999.9999
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **CLIMB OR UP-CUT** Q15:
  Climb milling: Input value = +1
  Conventional up-cut milling: Input value = –1
  Climb milling and up-cut milling alternately in several infeeds: Input value = 0

NC blocks

62 CYCL DEF 25 CONTOUR TRAIN	
Q1=-20	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q5=+0	;SURFACE COORDINATE
Q7=+50	;CLEARANCE HEIGHT
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q15=-1	;CLIMB OR UP-CUT

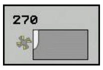
7.10 CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)

Please note while programming:

You can use this cycle to specify various properties of Cycle 25 CONTOUR TRAIN.

> Cycle 270 is DEF active, which means that it becomes effective as soon as it is defined in the part program.
>
> If Cycle 270 is used, do not define any radius compensation in the contour subprogram.
>
> Define Cycle 270 before Cycle 25.

Cycle parameters



- **Type of approach/departure (1/2/3)** Q390: Definition of the type of approach and departure:
  Q390=1:
  Approach the contour tangentially on a circular path
  Q390=2:
  Approach the contour tangentially on a straight line
  Q390=3:
  Approach the contour at a right angle
- **Radius compensation (0=R0/1=RL/2=RR)** Q391: Definition of the radius compensation:
  Q391=0:
  Machine the defined contour without radius compensation
  Q391=1:
  Machine the defined contour with radius compensation RL
  Q391=2:
  Machine the defined contour with radius compensation RR
- **Approach/departure radius** Q392: Only in effect if tangential approach on a circular path is selected (Q390=1). Radius of the approach/departure arc. Input range 0 to 99999.9999
- **Center angle** Q393: Only in effect if tangential approach on a circular path is selected (Q390=1). Angular length of the approach arc. Input range 0 to 99999.9999
- **Distance to auxiliary point** Q394: Only in effect if tangential approach on a straight line or right-angle approach is selected (Q390=2 or Q390=3). Distance to the auxiliary point from which the TNC is to approach the contour. Input range 0 to 99999.9999

NC blocks

```
62 CYCL DEF 270 CONTOUR TRAIN DATA
   Q390=1      ;TYPE OF APPROACH
   Q391=1      ;RADIUS COMPENSATION
   Q392=3      ;RADIUS
   Q393=+45    ;CENTER ANGLE
   Q394=+2     ;DISTANCE
```

7.11 TROCHOIDAL SLOT (Cycle 275, DIN ISO G275)

Cycle run

In conjunction with Cycle 14 **CONTOUR GEOMETRY,** this cycle facilitates the complete machining of open and closed slots or contour slots using trochoidal milling.

With trochoidal milling, large cutting depths and high cutting speeds are possible because the equally distributed cutting conditions prevent wear-increasing influences on the tool. When tool inserts are used the entire cutting length is exploited to increase the attainable chip volume per tooth. Moreover, trochoidal milling is easy on the machine mechanics. Enormous amounts of time can also be saved by combining this milling method with the integrated adaptive feed control **AFC** software option (see User's Manual on conversational programming).

Depending on the cycle parameters you select, the following machining alternatives are available:

- Complete machining: Roughing, side finishing
- Only roughing
- Only side finishing

Roughing with closed slots

The contour description of a closed slot must always start with a straight-line block (**L** block).

1 Following the positioning logic, the tool moves to the starting point of the contour description and moves in a reciprocating motion at the plunging angle defined in the tool table to the first plunging depth. Specify the plunging strategy with parameter **Q366**.

2 The TNC roughs the slot in circular motions to the contour end point. During the circular motion the TNC moves the tool in machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.

3 At the contour end point, the TNC moves the tool to clearance height and returns to the starting point of the contour description.

4 This process is repeated until the programmed slot depth is reached.

Finishing with closed slots

5 Inasmuch as a finishing allowance is defined, the TNC finishes the slot walls, in multiple infeeds if so specified. Starting from the defined starting point, the TNC approaches the slot wall tangentially. Climb or up-cut are taken into consideration.

Program structure: Machining with SL cycles

0 BEGIN PGM CYC275 MM
...
12 CYCL DEF 14.0 CONTOUR
13 CYCL DEF 14.1 CONTOUR LABEL 10
14 CYCL DEF 275 TROCHOIDAL SLOT...
15 CYCL CALL M3
...
50 L Z+250 R0 FMAX M2
51 LBL 10
...
55 LBL 0
...
99 END PGM CYC275 MM

Roughing with open slots

The contour description of an open slot must always start with an approach block (**APPR**).

1 Following the positioning logic, the tool moves to the starting point of the machining operation as defined by the parameters in the **APPR** block and positions there perpendicular to the first plunging depth.
2 The TNC roughs the slot in circular motions to the contour end point. During the circular motion the TNC moves the tool in machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the TNC moves the tool to clearance height and returns to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached.

Finishing with open slots

5 Inasmuch as a finishing allowance is defined, the TNC finishes the slot walls, in multiple infeeds if so specified. Starting from the defined starting point of the **APPR** block, the TNC approaches the slot wall. Climb or up-cut are taken into consideration.

Please note while programming:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

When using Cycle 275 TROCHOIDAL SLOT, you can define only one contour subprogram in Cycle 14 CONTOUR GEOMETRY.

Define the center line of the slot with all available path functions in the contour subprogram.

The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.

The TNC does not need Cycle 20 CONTOUR DATA in conjunction with Cycle 275.

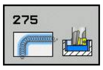
The starting point of a closed slot must not be located in a contour corner.

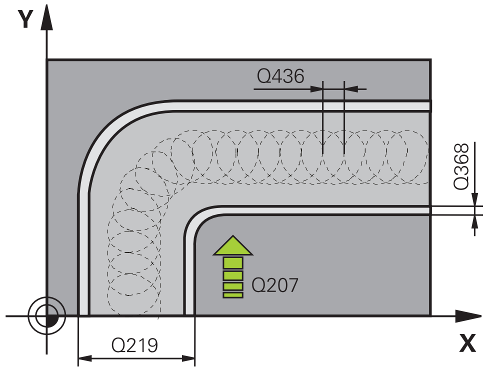
Danger of collision!

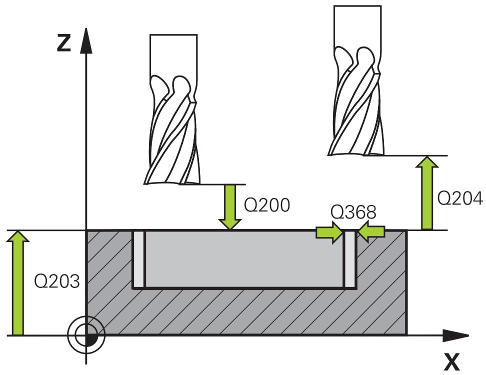
To avoid collisions,

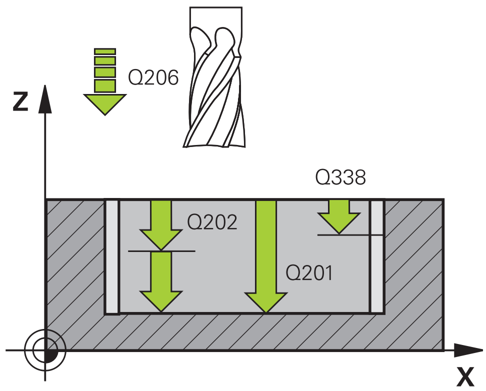
- Do not program positions in incremental dimensions immediately after Cycle 275 since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all principal axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.

Cycle parameters









- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Slot width** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Infeed per rev.** Q436 absolute: Value by which the TNC moves the tool in the machining direction per revolution. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999

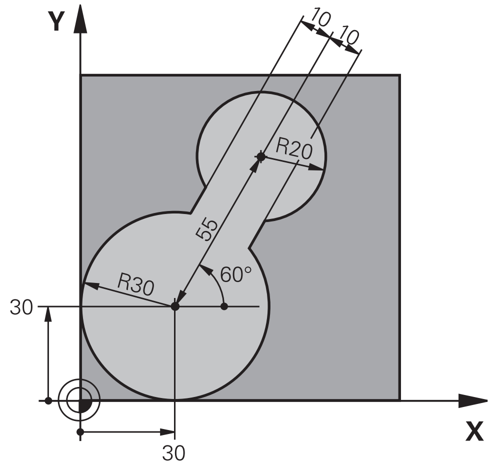
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Plunging strategy** Q366: Type of plunging strategy:
  0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle ANGLE defined in the tool table
  1 = No function
  2 = reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message
  Alternatively **PREDEF**

NC blocks

8 CYCL DEF 275 TROCHOIDAL SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q436=2	;INFEED PER REV.
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q385=500	;FINISHING FEED RATE
Q200=2	;SET-UP CLEARANCE
Q202=5	;PLUNGING DEPTH
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=2	;PLUNGE
9 CYCL CALL FMAX M3	

7.12 Programming Examples

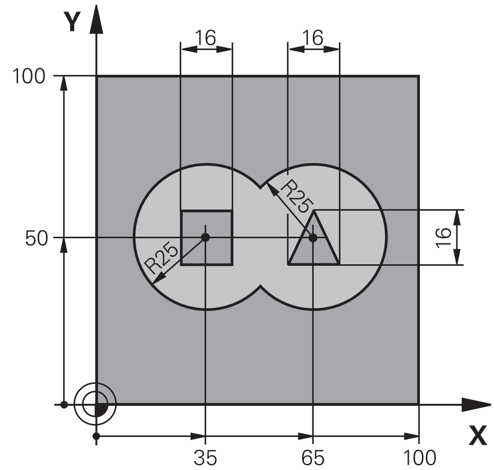
Example: Roughing-out and fine-roughing a pocket



0 BEGIN PGM C20 MM	
1 BLK FORM 0.1 Z X-10 Y-10 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	Definition of workpiece blank
3 TOOL CALL 1 Z S2500	Tool call: coarse roughing tool, diameter 30
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION	
8 CYCL DEF 22 ROUGH-OUT	Cycle definition: Coarse roughing
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR ROUGH-OUT	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
Q208=30000 ;RETRACTION FEED RATE	
9 CYCL CALL M3	Cycle call: Coarse roughing
10 L Z+250 R0 FMAX M6	Tool change

11 TOOL CALL 2 Z S3000	Tool call: fine roughing tool, diameter 15
12 CYCL DEF 22 ROUGH-OUT	Define the fine roughing cycle
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR ROUGH-OUT	
Q18=1 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
Q208=30000 ;RETRACTION FEED RATE	
13 CYCL CALL M3	Cycle call: Fine roughing
14 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
15 LBL 1	Contour subprogram
16 L X+0 Y+30 RR	
17 FC DR- R30 CCX+30 CCY+30	
18 FL AN+60 PDX+30 PDY+30 D10	
19 FSELECT 3	
20 FPOL X+30 Y+30	
21 FC DR- R20 CCPR+55 CCPA+60	
22 FSELECT 2	
23 FL AN-120 PDX+30 PDY+30 D10	
24 FSELECT 3	
25 FC X+0 DR- R30 CCX+30 CCY+30	
26 FSELECT 2	
27 LBL 0	
28 END PGM C20 MM	

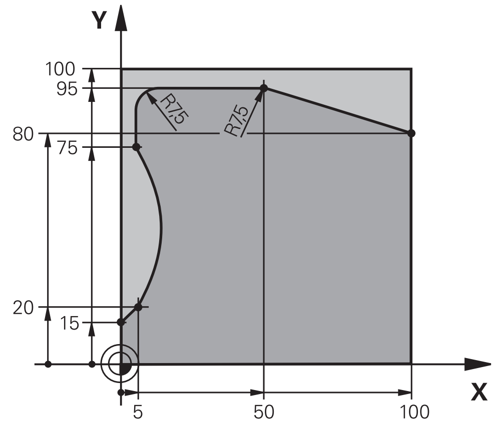
Example: Pilot drilling, roughing-out and finishing overlapping contours



0 BEGIN PGM C21 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 1 Z S2500	Tool call: Drill, diameter 12
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1 /2 /3 /4	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION	
8 CYCL DEF 21 PILOT DRILLING	Cycle definition: Pilot drilling
Q10=5 ;PLUNGING DEPTH	
Q11=250 ;FEED RATE FOR PLNGNG	
Q13=2 ;ROUGH-OUT TOOL	
9 CYCL CALL M3	Cycle call: Pilot drilling
10 L +250 R0 FMAX M6	Tool change
11 TOOL CALL 2 Z S3000	Call the tool for roughing/finishing, diameter 12
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: Rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR ROUGH-OUT	

Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
Q208=30000 ;RETRACTION FEED RATE	
13 CYCL CALL M3	Cycle call: Rough-out
14 CYCL DEF 23 FLOOR FINISHING	Cycle definition: Floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR MILLING	
Q208=30000 ;RETRACTION FEED RATE	
15 CYCL CALL	Cycle call: Floor finishing
16 CYCL DEF 24 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;DIRECTION OF ROTATION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE FOR MILLING	
Q14=+0 ;ALLOWANCE FOR SIDE	
17 CYCL CALL	Cycle call: Side finishing
18 L Z+250 R0 FMAX M2	Retract the tool, end program
19 LBL 1	Contour subprogram 1: left pocket
20 CC X+35 Y+50	
21 L X+10 Y+50 RR	
22 C X+10 DR-	
23 LBL 0	
24 LBL 2	Contour subprogram 2: right pocket
25 CC X+65 Y+50	
26 L X+90 Y+50 RR	
27 C X+90 DR-	
28 LBL 0	
29 LBL 3	Contour subprogram 3: square left island
30 L X+27 Y+50 RL	
31 L Y+58	
32 L X+43	
33 L Y+42	
34 L X+27	
35 LBL 0	
36 LBL 4	Contour subprogram 4: triangular right island
37 L X+65 Y+42 RL	
38 L X+57	
39 L X+65 Y+58	
40 L X+73 Y+42	
41 LBL 0	
42 END PGM C21 MM	

Example: Contour train



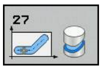
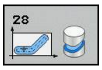
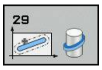
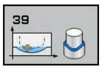
0 BEGIN PGM C25 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2000	Tool call: Diameter 20
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 25 CONTOUR TRAIN	Define machining parameters
Q1=-20 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q5=+0 ;SURFACE COORDINATE	
Q7=+250 ;CLEARANCE HEIGHT	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR MILLING	
Q15=+1 ;CLIMB OR UP-CUT	
8 CYCL CALL M3	Cycle call
9 L Z+250 R0 FMAX M2	Retract the tool, end program
10 LBL 1	Contour subprogram
11 L X+0 Y+15 RL	
12 L X+5 Y+20	
13 CT X+5 Y+75	
14 L Y+95	
15 RND R7.5	
16 L X+50	
17 RND R7.5	
18 L X+100 Y+80	
19 LBL 0	
20 END PGM C25 MM	

8

Fixed Cycles: Cylindrical Surface

8.1 Fundamentals

Overview of cylindrical surface cycles

Cycle	Soft key	Page
27 CYLINDER SURFACE		223
28 CYLINDER SURFACE slot milling		226
29 CYLINDER SURFACE ridge milling		229
39 CYLINDER SURFACE Contour		232

8.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, software option 1)

Cycle run

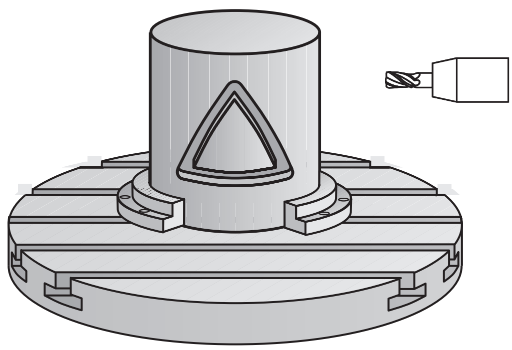
This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle 28 if you want to mill guideways on the cylinder.

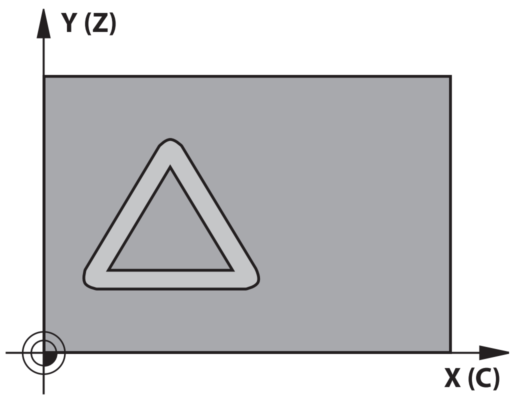
The contour is described in a subprogram identified in Cycle 14 CONTOUR GEOMETRY.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

The dimensions for the rotary axis (X coordinates) can be entered as desired either in degrees or in mm (or inches). Specify this with Q17 in the cycle definition.

1 The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.
3 At the end of the contour, the TNC returns the tool to the set-up clearance and returns to the point of penetration.
4 Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.
5 Then the tool moves to the set-up clearance.





Please note while programming:

The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder.

Refer to your machine manual.

In the first NC block of the contour program, always program both cylinder surface coordinates.

The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table. Set the reference point to the center of the rotary table.

The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the TNC will generate an error message. Switching of the kinematics may be required.

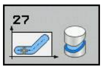
This cycle can also be used in a tilted working plane.

The set-up clearance must be greater than the tool radius.

The machining time can increase if the contour consists of many non-tangential contour elements.

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? deg=0 MM/INCH=1** Q17: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

NC blocks

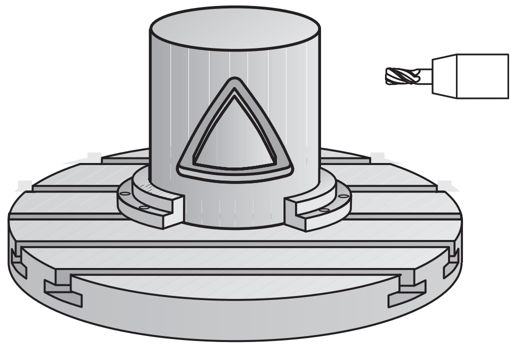
63 CYCL DEF 27 CYLINDER SURFACE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q16=25	;RADIUS
Q17=0	;DIMENSION TYPE

8.3 CYLINDER SURFACE Slot milling (Cycle 28, DIN/ISO: G128, software option 1)

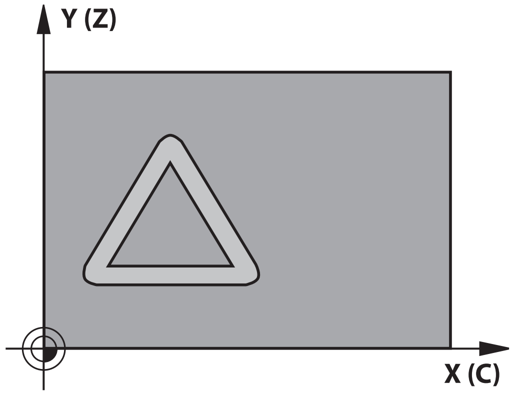
Cycle run

With this cycle you can program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle 27, this cycle enables the TNC to adjust the tool so that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel walls by using a tool that is exactly as wide as the slot.

The smaller the tool is with respect to the slot width, the larger the distortion in circular arcs and oblique line segments. To minimize this process-related distortion, you can define the parameter Q21. This parameter specifies the tolerance with which the TNC machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.



Program the midpoint path of the contour together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the slot with climb milling or up-cut milling.



1 The TNC positions the tool over the cutter infeed point.
2 The TNC moves the tool to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate Q12. The approaching behavior depends on the parameter ConfigDatum CfgGeoCycle apprDepCylWall.
3 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate Q12 while respecting the finishing allowance for the side.
4 At the end of the contour, the TNC moves the tool to the opposite wall and returns to the infeed point.
5 Steps 2 to 3 are repeated until the programmed milling depth Q1 is reached.
6 If you have defined the tolerance in Q21, the TNC then remachines the slot walls to be as parallel as possible.
7 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.

Please note while programming:

This cycle performs an inclined 5-axis machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

Define the approaching behavior in ConfigDatum, CfgGeoCycle, apprDepCylWall

- CircleTangential:
  Tangential approach and departure
- LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

In the first NC block of the contour program, always program both cylinder surface coordinates.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table. Set the reference point to the center of the rotary table.

The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

This cycle can also be used in a tilted working plane.

The set-up clearance must be greater than the tool radius.

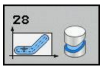
The machining time can increase if the contour consists of many non-tangential contour elements.

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

At the end of the cycle, move the tool in the plane to an absolute position, not to an incremental position, if you have set the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket to ToolAxClearanceHeight.

In the parameter CfgGeoCycle, displaySpindleErr, on/off, define whether the TNC should output an error message (on) or not (off) if spindle rotation is not active when the cycle is called. The function needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? deg=0 MM/INCH=1** Q17: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- **Slot width** Q20: Width of the slot to be machined. Input range -99999.9999 to 99999.9999
- **Tolerance** Q21: If you use a tool smaller than the programmed slot width Q20, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance Q21, the TNC adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With Q21 you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the remachining takes. Input range for tolerance 0.0001 to 9.9999
  Recommendation: Use a tolerance of 0.02 mm.
  Function inactive: Enter 0 (default setting)

NC blocks

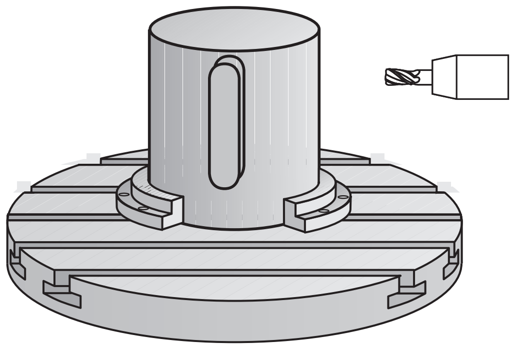
63 CYCL DEF 28 CYLINDER SURFACE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q16=25	;RADIUS
Q17=0	;DIMENSION TYPE
Q20=12	;SLOT WIDTH
Q21=0	;TOLERANCE

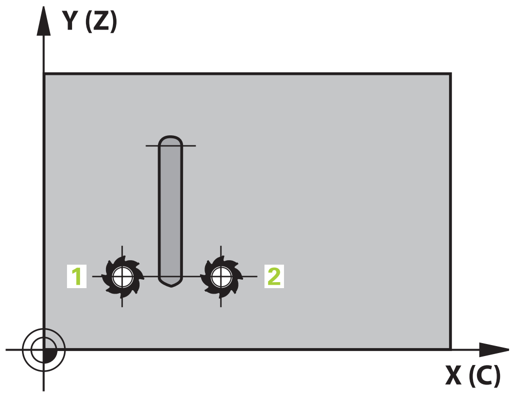
8.4 CYLINDER SURFACE Ridge milling (Cycle 29, DIN/ISO: G129, software option 1)

Cycle run

This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle the TNC adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the midpoint path of the ridge together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge the TNC always adds a semicircle whose radius is half the ridge width.





1 The TNC positions the tool over the starting point of machining. The TNC calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins from the left (**1**, RL = climb milling) or the right of the ridge (**2**, RR = up-cut milling).
2 After the TNC has positioned to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the ridge wall. If so programmed, it will leave metal for the finishing allowance.
3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate Q12 until the stud is completed.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle.

Please note while programming:

This cycle performs an inclined 5-axis machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

In the first NC block of the contour program, always program both cylinder surface coordinates.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table. Set the reference point to the center of the rotary table.

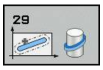
The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the TNC will generate an error message. Switching of the kinematics may be required.

The set-up clearance must be greater than the tool radius.

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

In the parameter CfgGeoCycle, displaySpindleErr, on/off, define whether the TNC should output an error message (on) or not (off) if spindle rotation is not active when the cycle is called. The function needs to be adapted by your machine manufacturer.

Cycle parameters



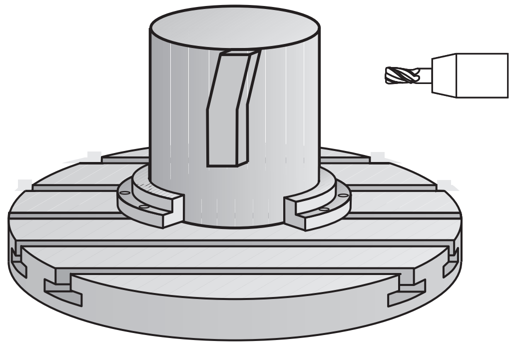
- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? deg=0 MM/INCH=1** Q17: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- **Ridge width** Q20: Width of the ridge to be machined. Input range -99999.9999 to 99999.9999

NC blocks

```
63 CYCL DEF 29 CYLINDER SURFACE
   RIDGE
   Q1=-8      ;MILLING DEPTH
   Q3=+0      ;ALLOWANCE FOR SIDE
   Q6=+0      ;SET-UP CLEARANCE
   Q10=+3     ;PLUNGING DEPTH
   Q11=100    ;FEED RATE FOR
              PLNGNG
   Q12=350    ;FEED RATE FOR
              MILLING
   Q16=25     ;RADIUS
   Q17=0      ;DIMENSION TYPE
   Q20=12     ;RIDGE WIDTH
```

8.5 CYLINDER SURFACE (Cycle 39, DIN/ISO: G139, software option 1)

Cycle run



This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle the TNC adjusts the tool so that, with radius compensation active, the wall of the open contour is always parallel to the cylinder axis.

The contour is described in a subprogram identified in Cycle 14 CONTOUR GEOMETRY.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

Unlike Cycles 28 and 29, in the contour subprogram you define the actual contour to be machined.

1 The TNC positions the tool over the starting point of machining. The TNC locates the starting point next to the first point defined in the contour subprogram, offset by the tool diameter.
2 The TNC then moves the tool to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate Q12. A finishing allowance programmed for the side is taken into account. (The approaching behavior depends on the parameter ConfigDatum, CfgGeoCycle, apprDepCylWall.)
3 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12 until the contour train is completed.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket).

Please note while programming:

This cycle performs an inclined 5-axis machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

In the first NC block of the contour program, always program both cylinder surface coordinates.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Ensure that the tool has enough space laterally for contour approach and departure.

The cylinder must be set up centered on the rotary table. Set the reference point to the center of the rotary table.

The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

The set-up clearance must be greater than the tool radius.

The machining time can increase if the contour consists of many non-tangential contour elements.

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

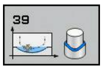
Define the approaching behavior in ConfigDatum, CfgGeoCycle, apprDepCylWall

- CircleTangential:
  Tangential approach and departure
- LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

Danger of collision!

In the parameter CfgGeoCycle, displaySpindleErr, on/off, define whether the TNC should output an error message (on) or not (off) if spindle rotation is not active when the cycle is called. The function needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? deg=0 MM/INCH=1** Q17: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

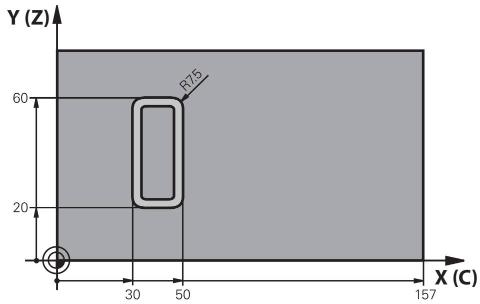
NC blocks

63 CYCL DEF 39 CYL. SURFACE CONTOUR	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q16=25	;RADIUS
Q17=0	;DIMENSION TYPE

8.6 Programming Examples

Example: Cylinder surface with Cycle 27

- Machine with B head and C table
- Cylinder centered on rotary table
- Datum is on the underside, in the center of the rotary table

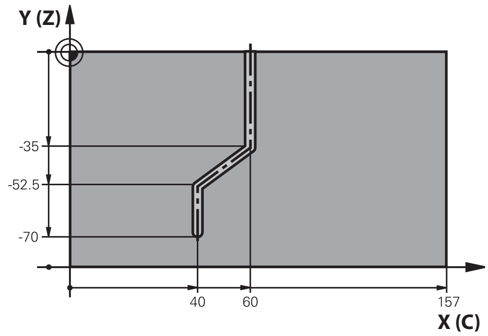


0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call: Diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position tool at rotary table center
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Positioning
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 27 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE FOR MILLING	
Q16=25 ;RADIUS	
Q17=1 ;DIMENSION TYPE	
8 L C+0 R0 FMAX M13 M99	Pre-position rotary table, spindle ON, call the cycle
9 L Z+250 R0 FMAX	Retract the tool
10 PLANE RESET TURN FMAX	Tilt back, cancel the PLANE function
11 M2	End of program
12 LBL 1	Contour subprogram
13 L X+40 Y+20 RL	Data for the rotary axis are entered in mm (Q17=1)
14 L X+50	
15 RND R7.5	
16 L Y+60	
17 RND R7.5	
18 L IX-20	
19 RND R7.5	
20 L Y+20	

21 RND R7.5	
22 L X+40 Y+20	
23 LBL 0	
24 END PGM C27 MM	

Example: Cylinder surface with Cycle 28

- Cylinder centered on rotary table
- Machine with B head and C table
- Datum at center of rotary table
- Description of the midpoint path in the contour subprogram



0 BEGIN PGM C28 MM	
1 TOOL CALL 1 Z S2000	Tool call, tool axis Z, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y+0 R0 FMAX	Position tool at rotary table center
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN FMAX	Tilting
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 28 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=-4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE FOR MILLING	
Q16=25 ;RADIUS	
Q17=1 ;DIMENSION TYPE	
Q20=10 ;SLOT WIDTH	
Q21=0.02 ;TOLERANCE	Remachining active
8 L C+0 R0 FMAX M3 M99	Pre-position rotary table, spindle ON, call the cycle
9 L Z+250 R0 FMAX	Retract the tool
10 PLANE RESET TURN FMAX	Tilt back, cancel the PLANE function
11 M2	End of program
12 LBL 1	Contour subprogram, description of the midpoint path
13 L X+60 Y+0 RL	Data for the rotary axis are entered in mm (Q17=1)
14 L Y-35	
15 L X+40 Y-52.5	
16 L Y-70	
17 LBL 0	
18 END PGM C28 MM	

9

Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

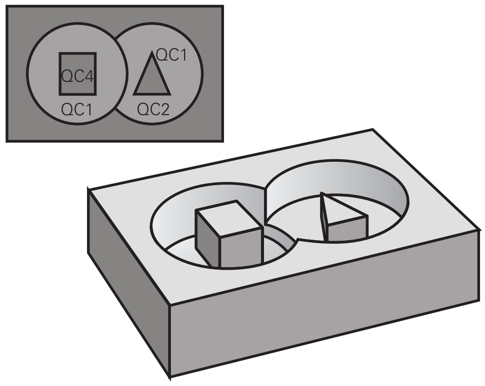
Fundamentals

SL cycles and the complex contour formula enable you to form complex contours by combining subcontours (pockets or islands). You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the complete contour from the selected subcontours, which you link together through a contour formula.

The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** elements.

The SL cycles with contour formula presuppose a structured program layout and enable you to save frequently used contours in individual programs. Using the contour formula, you can connect the subcontours to a complete contour and define whether it applies to a pocket or island.

In its present form, the "SL cycles with contour formula" function requires input from several areas in the TNC's user interface. This function is to serve as a basis for further development.



Program structure: Machining with SL cycles and complex contour formula

0 BEGIN PGM CONTOUR MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA...
8 CYCL DEF 22 ROUGH-OUT...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTOUR MM

Properties of the subcontours

- By default, the TNC assumes that the contour is a pocket. Do not program a radius compensation.
- The TNC ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram.
- You can define subcontours with various depths as needed

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies to the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN PGM MODEL MM
1 DECLARE CONTOUR QC1 = "CIRCLE1"
2 DECLARE CONTOUR QC2 =
    "CIRCLEXY" DEPTH15
3 DECLARE CONTOUR QC3 =
    "TRIANGLE" DEPTH10
4 DECLARE CONTOUR QC4 = "SQUARE"
    DEPTH5
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM MODEL MM

0 BEGIN PGM CIRCLE 1 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM CIRCLE 1 MM

0 BEGIN PGM CIRCLE31XY MM
...
...
```

Selecting a program with contour definitions

With the **SEL CONTOUR** function you select a program with contour definitions, from which the TNC takes the contour descriptions:

SPEC FCT

- Show the soft-key row with special functions

CONTOUR + POINT MACHINING

- Select the menu for functions for contour and point machining

SEL CONTOUR

- Press the **SEL CONTOUR** soft key
- Enter the full name of the program with the contour definition and confirm with the **END** key

Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR GEOMETRY** is no longer necessary if you use **SEL CONTOUR**.

Defining contour descriptions

With the **DECLARE CONTOUR** function you enter in a program the path for programs from which the TNC draws the contour descriptions. In addition, you can select a separate depth for this contour description (FCL 2 function):

SPEC FCT

- Show the soft-key row with special functions

CONTOUR + POINT MACHINING

- Select the menu for functions for contour and point machining

DECLARE CONTOUR

- Press the **DECLARE CONTOUR** soft key
- Enter the number for the contour designator **QC**, and confirm with the **ENT** key
- Enter the full name of the program with the contour description and confirm with the **END** key, or if desired,
- Define a separate depth for the selected contour

With the entered contour designators **QC** you can include the various contours in the contour formula.

If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).

Entering a complex contour formula

You can use soft keys to interlink various contours in a mathematical formula.

SPEC FCT

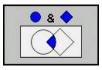
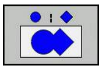
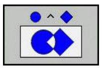
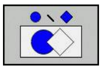
- Show the soft-key row with special functions

CONTOUR + POINT MACHINING

- Select the menu for functions for contour and point machining

CONTOUR FORMULA

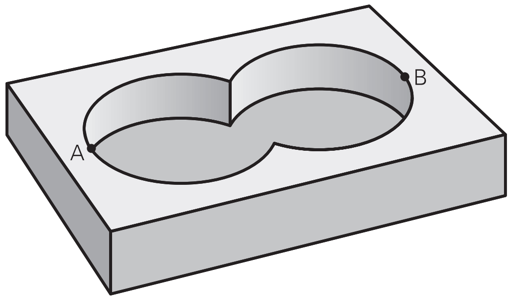
- Press the **CONTOUR FORMULA** soft key. The TNC then displays the following soft keys:

Mathematical function	Soft key
cut with e.g. **QC10 = QC1 & QC5**	
joined with e.g. **QC25 = QC7 | QC18**	
joined with, but without cut e.g. **QC12 = QC5 ^ QC25**	
without e.g. **QC25 = QC1 \ QC2**	
Parenthesis open e.g. **QC12 = QC1 * (QC2 + QC3)**	(
Parenthesis closed e.g. **QC12 = QC1 * (QC2 + QC3)**	)
Define single contour e.g. **QC12 = QC1**	

Superimposed contours

By default, the TNC considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



Subprograms: overlapping pockets

The following programming examples are contour description programs that are defined in a contour definition program. The contour definition program is called through the **SEL CONTOUR** function in the actual main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Contour description program 1: pocket A

```
0 BEGIN PGM POCKET_A MM
1 L X+10 Y+50 R0
2 CC X+35 Y+50
3 C X+10 Y+50 DR-
4 END PGM POCKET_A MM
```

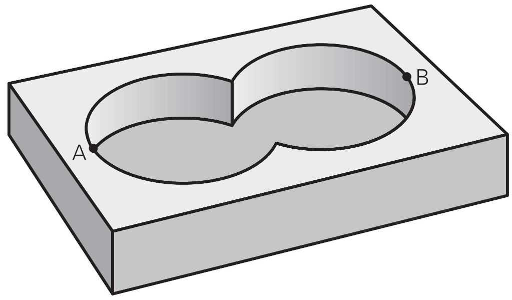
Contour description program 2: pocket B

```
0 BEGIN PGM POCKET_B MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET_B MM
```

Area of inclusion

Both areas A and B are to be machined, including the overlapping area:

- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.



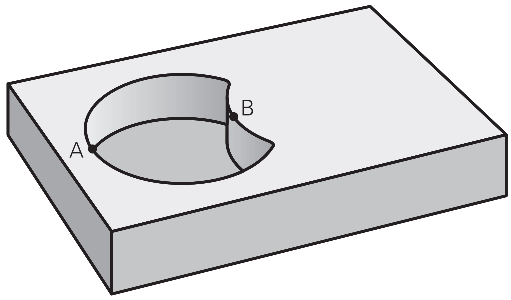
Contour definition program:

```
50 ...
51 ...
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 | QC2
55 ...
56 ...
```

Area of exclusion

Area A is to be machined without the portion overlapped by B:

- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the area B is subtracted from the area A with the **without** function.



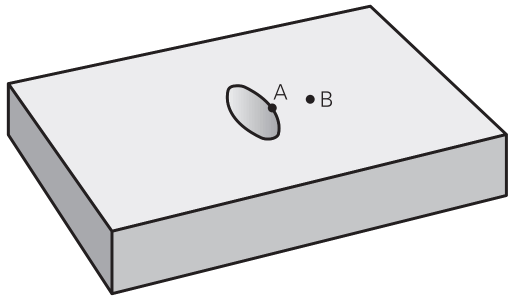
Contour definition program:

```
50 ...
51 ...
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 \ QC2
55 ...
56 ...
```

Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "intersection with" function.



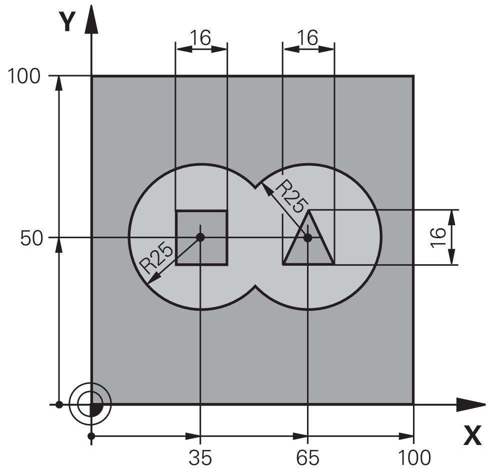
Contour definition program:

```
50 ...
51 ...
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 & QC2
55 ...
56 ...
```

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", page 189).

Example: Roughing and finishing superimposed contours with the contour formula



0 BEGIN PGM CONTOUR 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 DEF 1 L+0 R+2.5	Tool definition of roughing cutter
4 TOOL DEF 2 L+0 R+3	Tool definition of finishing cutter
5 TOOL CALL 1 Z S2500	Tool call of roughing cutter
6 L Z+250 R0 FMAX	Retract the tool
7 SEL CONTOUR "MODEL"	Specify contour definition program
8 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION	

9 CYCL DEF 22 ROUGH-OUT	Cycle definition: Rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR MILLING	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
Q401=100 ;FEED RATE FACTOR	
Q404=0 ;FINE ROUGH STRATEGY	
10 CYCL CALL M3	Cycle call: Rough-out
11 TOOL CALL 2 Z S5000	Tool call of finishing cutter
12 CYCL DEF 23 FLOOR FINISHING	Cycle definition: Floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR MILLING	
13 CYCL CALL M3	Cycle call: Floor finishing
14 CYCL DEF 24 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;DIRECTION OF ROTATION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE FOR MILLING	
Q14=+0 ;ALLOWANCE FOR SIDE	
15 CYCL CALL M3	Cycle call: Side finishing
16 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
17 END PGM CONTOUR MM	

Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	Contour definition program
1 DECLARE CONTOUR QC1 = "CIRCLE1"	Definition of the contour designator for the program "CIRCLE1"
2 FN 0: Q1 =+35	Assignment of values for parameters used in PGM "CIRCLE31XY"
3 FN 0: Q2 =+50	
4 FN 0: Q3 =+25	
5 DECLARE CONTOUR QC2 = "CIRCLE31XY"	Definition of the contour designator for the program "CIRCLE31XY"
6 DECLARE CONTOUR QC3 = "TRIANGLE"	Definition of the contour designator for the program "TRIANGLE"
7 DECLARE CONTOUR QC4 = "SQUARE"	Definition of the contour designator for the program "SQUARE"
8 QC10 = (QC 1 | QC 2) \ QC 3 \ QC 4	Contour formula
9 END PGM MODEL MM	

Contour description programs:

0 BEGIN PGM CIRCLE 1 MM	Contour description program: circle at right
1 CC X+65 Y+50	
2 L PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE 1 MM	

0 BEGIN PGM CIRCLE31XY MM	Contour description program: circle at left
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE31XY MM	

0 BEGIN PGM TRIANGLE MM	Contour description program: triangle at right
1 L X+73 Y+42 R0	
2 L X+65 Y+58	
3 L X+58 Y+42	
4 L X+73	
5 END PGM TRIANGLE MM	

0 BEGIN PGM SQUARE MM	Contour description program: square at left
1 L X+27 Y+58 R0	
2 L X+43	
3 L Y+42	
4 L X+27	
5 L Y+58	
6 END PGM SQUARE MM	

9.2 SL cycles with simple contour formula

Fundamentals

SL cycles and the simple contour formula enable you to form contours by combining up to 9 subcontours (pockets or islands) in a simple manner. You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the contour from the selected subcontours.

> The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** elements.

Program structure: Machining with SL cycles and complex contour formula

0 BEGIN PGM CONTDEF MM
...
5 CONTOUR DEF P1= "POCK1.H" I2 = "ISLE2.H" DEPTH5 I3 "ISLE3.H" DEPTH7.5
6 CYCL DEF 20 CONTOUR DATA...
8 CYCL DEF 22 ROUGH-OUT...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTDEF MM

Properties of the subcontours

- Do not program a radius compensation.
- The TNC ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram.

Characteristics of the fixed cycles

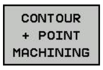
- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies to the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Entering a simple contour formula

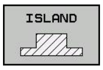
You can use soft keys to interlink various contours in a mathematical formula.

- Show the soft-key row with special functions



- Select the menu for functions for contour and point machining

- Press the **CONTOUR DEF** soft key. The TNC opens the dialog for entering the contour formula
- Enter the name of the first subcontour. The first subcontour must always be the deepest pocket. Confirm with the **ENT** key



- Specify via soft key whether the next subcontour is a pocket or an island. Confirm with the **ENT** key
- Enter the name of the second subcontour. Confirm with the **ENT** key
- If needed, enter the depth of the second subcontour. Confirm with the **ENT** key
- Carry on with the dialog as described above until you have entered all subcontours.

Always start the list of subcontours with the deepest pocket!

If the contour is defined as an island, the TNC interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!

If the depth is entered as 0, then for pockets the depth defined in the Cycle 20 is effective. Islands then rise up to the workpiece top surface!

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", page 189).

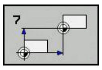
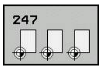
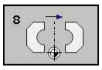
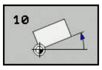
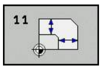
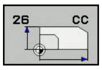
10

Cycles: Coordinate Transformations

10.1 Fundamentals

Overview

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The TNC provides the following coordinate transformation cycles:

Cycle	Soft key	Page
7 DATUM For shifting contours directly within the program or from datum tables		255
247 DATUM SETTING Datum setting during program run		261
8 MIRRORING Mirroring contours		262
10 ROTATION Rotating contours in the working plane		264
11 SCALING FACTOR Increasing or reducing the size of contours		266
26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling		267
19 WORKING PLANE Machining in tilted coordinate system on machines with swivel heads and/or rotary tables		269

Effect of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

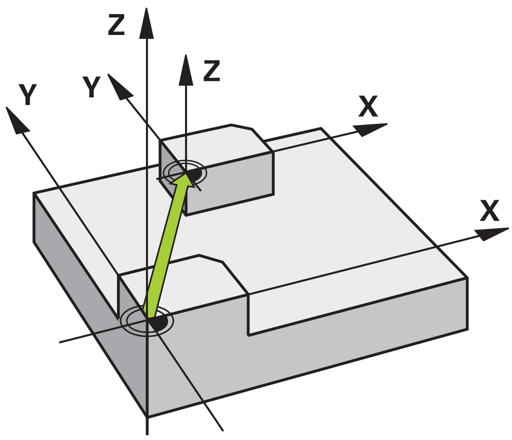
- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM block (depending on machine parameter **clearMode**).
- Select a new program

10.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54)

Effect

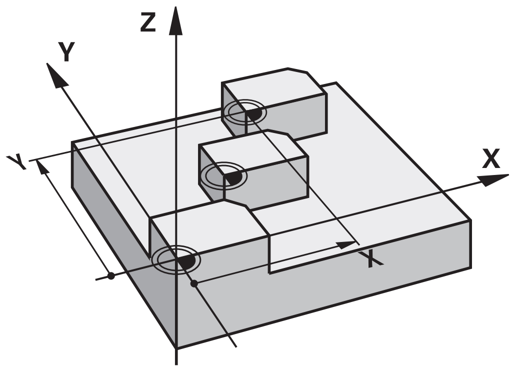
A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.

When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

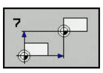


Resetting

- Program a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.



Cycle parameters



- **Datum shift**: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to six NC axes, each from –99999.9999 to 99999.9999

NC blocks

13 CYCL DEF 7.0 DATUM
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 7.3 Z-5

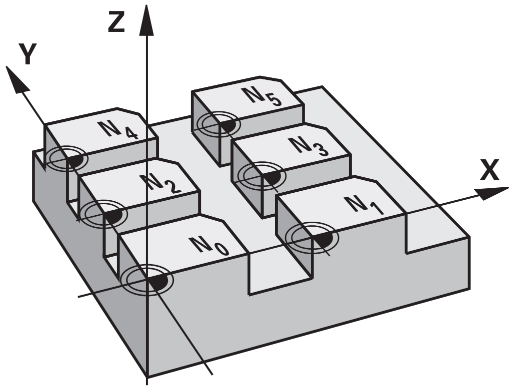
10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

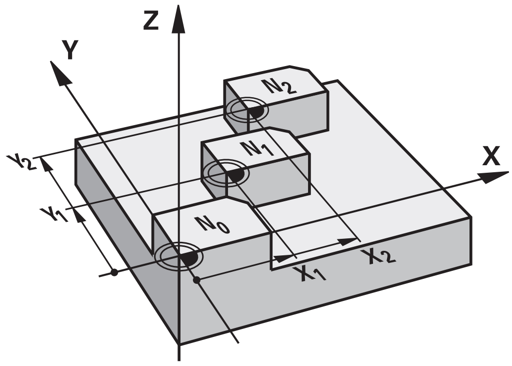
Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.





Resetting

- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
- Execute a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition

Status displays

In the additional status display, the following data from the datum table are shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

Please note while programming:

Danger of collision!

Datums from a datum table are **always and exclusively** referenced to the current datum (preset).

If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

If you work without **SEL TABLE**, then you must activate the desired datum table before the test run or the program run. (This applies also to the programming graphics).

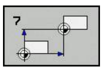
- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table receives the status S
- Use the file management in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes to select the desired table for program run: The table receives the status M

The coordinate values from datum tables are only effective with absolute coordinate values.

New lines can only be inserted at the end of the table.

If you create datum tables, the file name has to start with a letter.

Cycle parameters



- **Datum shift**: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number entered in the Q parameter. Input range: 0 to 9999

NC blocks

```
77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1 #5
```

Selecting a datum table in the part program

With the **SEL TABLE** function you select the table from which the TNC takes the datums:

PGM CALL

- Select the functions for program call: Press the **PGM CALL** key

DATUM TABLE

- Press the **DATUM TABLE** soft key
- Select the complete path name of the datum table or the file with the **SELECT** soft key and confirm your entry with the **END** key

> Program a **SEL TABLE** block before Cycle 7 Datum Shift.
>
> A datum table selected with **SEL TABLE** remains active until you select another datum table with **SEL TABLE** or through **PGM MGT**.

Edit the datum table in the Programming mode of operation

> After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise the change might not be included during program run.

Select the datum table in the **Programming** mode of operation

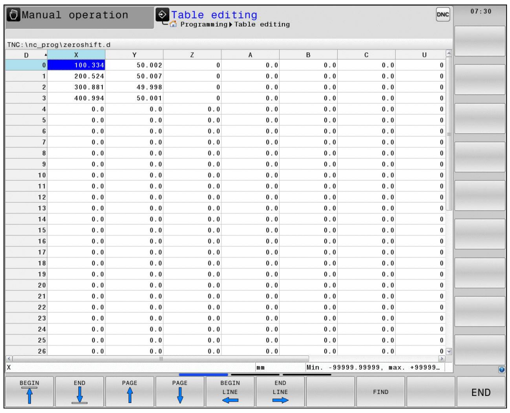
PGM MGT

- Call the file manager: Press the **PGM MGT** key
- Display the datum tables: Press the **SELECT TYPE** and **SHOW .D** soft keys
- Select the desired table or enter a new file name.
- Edit the file The functions displayed in the soft-key row for editing include:

Function	Soft key
Select beginning of table	BEGIN ↑
Select end of table	END ↓
Go to the previous page	PAGE ↑
Go to next page	PAGE ↓
Insert line (only possible at the end of table)	INSERT LINE
Delete line	DELETE LINE
Find	FIND
Go to beginning of line	BEGIN LINE ←
Go to end of line	END LINE →
Copy the current value	COPY FIELD
Insert the copied value	PASTE FIELD
Add the entered number of lines (datums) to the end of the table	APPEND N LINES

Configuring the datum table

If you do not wish to define a datum for an active axis, press the **DEL** key. Then the TNC clears the numerical value from the corresponding input field.



> You can change the properties of tables. Enter the code number 555343 in the MOD menu. The TNC then offers the EDIT FORMAT soft key if a table is selected. When you press this soft key, the TNC opens a pop-up window where the properties are shown for each column of the selected table. Any changes made only affect the open table.

To exit a datum table

Select a different type of file in file management and choose the desired file.

> After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise the change may not be included during program run.

Status displays

In the additional status display, the TNC shows the values of the active datum shift.

10.4 DATUM SETTING (Cycle 247, DIN/ ISO: G247)

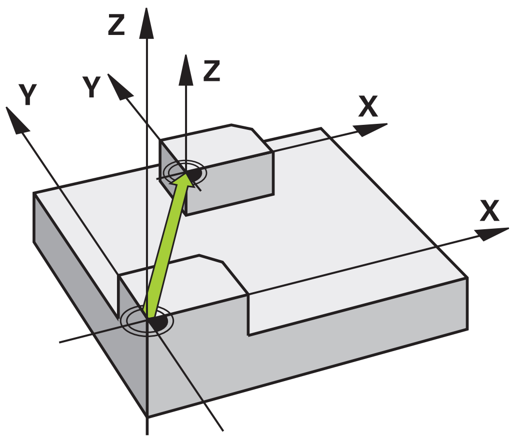
Effect

With the DATUM SETTING cycle you can activate as the new datum a preset defined in a preset table.

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display

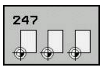
In the status display the TNC shows the active preset number behind the datum symbol.



Please note before programming:

> When activating a datum from the preset table, the TNC resets the datum shift, mirroring, rotation, scaling factor and axis-specific scaling factor.
>
> If you activate preset number 0 (line 0), then you activate the datum that you last set in the **Manual Operation** or **El. Handwheel** operating mode.
>
> Cycle 247 is not functional in **Test Run** mode.

Cycle parameters



- **Number for datum?**: Enter the number of the datum to be activated from the preset table. Input range: 0 to 65535

NC blocks

```
13 CYCL DEF 247 DATUM SETTING
   Q339=4     ;DATUM NUMBER
```

Status displays

In the additional status display (**POS. DISP. STATUS**) the TNC shows the active preset number behind the **datum** dialog.

10.5 MIRRORING (Cycle 8, DIN/ISO: G28)

Effect

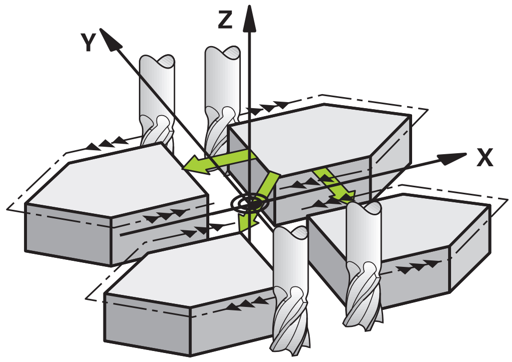
The TNC can machine the mirror image of a contour in the working plane.

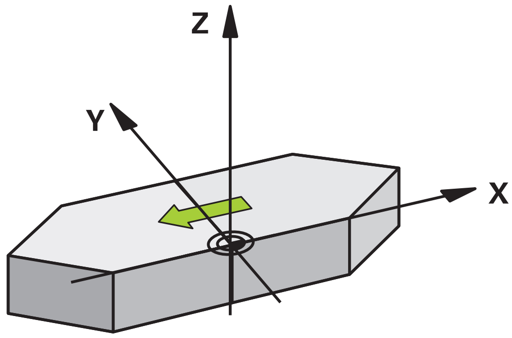
The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in the **Positioning with MDI** mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in SL cycles).
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.





Resetting

Program the MIRROR IMAGE cycle once again with **NO ENT**.

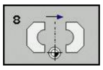
Please note while programming

When using Cycle 8 in a tilted system please remember:

- **First** program the tilting movement and **then** call Cycle 8 MIRRORING!

If you call Cycle 8 before you have tilted the working plane, the TNC will output an error message.

Cycle parameters



- **Mirrored axis?**: Enter the axis to be mirrored. You can mirror all axes except for the spindle axis—including rotary axes—with the exception of the spindle axis and its associated auxiliary axis. You can enter up to three axes. Input range: Up to three NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

NC blocks

```
79 CYCL DEF 8.0 MIRROR IMAGE
80 CYCL DEF 8.1 X Y Z
```

10.6 ROTATION (Cycle 10, DIN/ISO: G73)

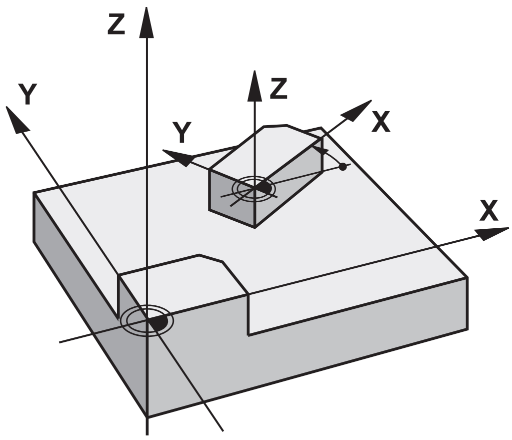
Effect

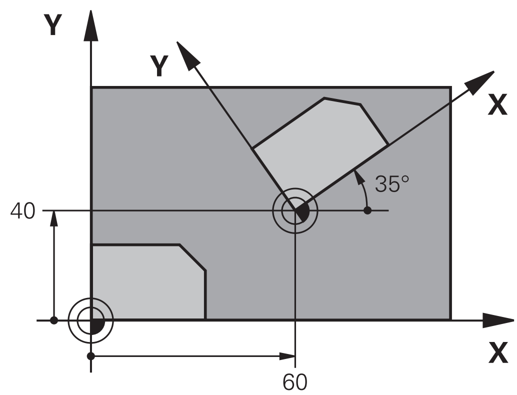
The TNC can rotate the coordinate system about the active datum in the working plane within a program.

The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis





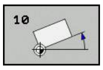
Resetting

Program the ROTATION cycle once again with a rotation angle of 0°.

Please note while programming:

> An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.
>
> After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

Cycle parameters



- **Rotation**: Enter the rotation angle in degrees (°). Input range –360.000° to +360.000° (absolute or incremental)

NC blocks

```
12 CALL LBL 1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 ROTATION
17 CYCL DEF 10.1 ROT+35
18 CALL LBL 1
```

10.7 SCALING (Cycle 11, DIN/ISO: G72

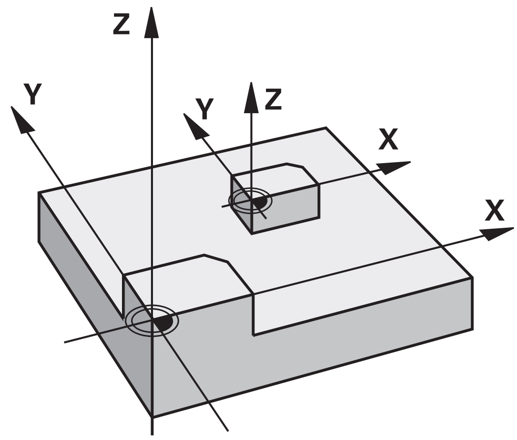
Effect

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the **Positioning with MDI** mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- all three coordinate axes at the same time
- Dimensions in cycles



Prerequisite

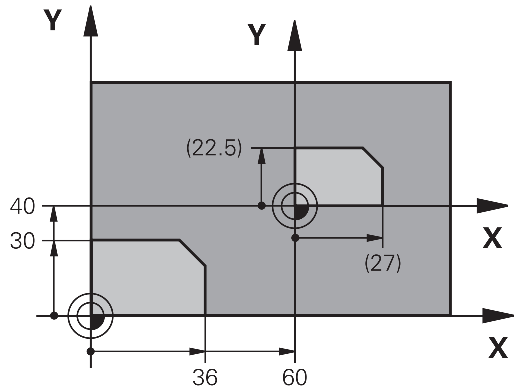
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

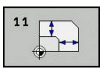
Reduction: SCL less than 1 (down to 0.000 001)

Resetting

Program the SCALING cycle once again with a scaling factor of 1.



Cycle parameters



- **Scaling factor?**: Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range 0.000001 to 99.999999

NC blocks

```
11 CALL LBL 1
12 CYCL DEF 7.0 DATUM
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1
```

10.8 AXIS-SPECIFIC SCALING (Cycle 26)

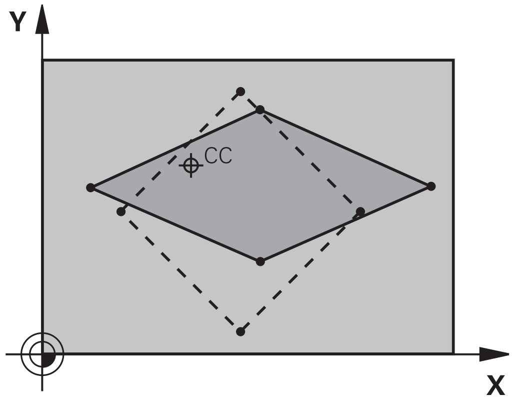
Effect

With Cycle 26 you can account for shrinkage and oversize factors for each axis.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the **Positioning with MDI** mode of operation. The active scaling factor is shown in the additional status display.

Resetting

Program the SCALING cycle once again with a scaling factor of 1 for the same axis.



Please note while programming:

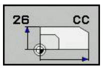
Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

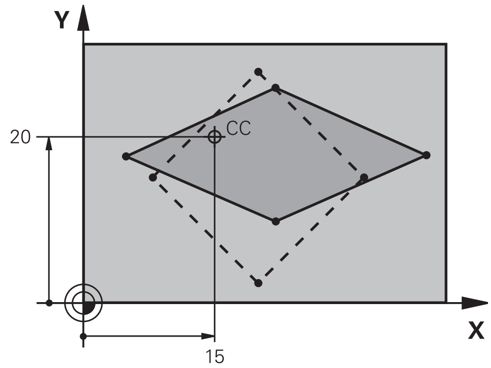
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters



- **Axis and scaling factor**: Select the coordinate axis/ axes by soft key and enter the factor(s) involved in enlarging or reducing. Input range 0.000001 to 99.999999
- **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range -99999.9999 to 99999.9999



NC blocks

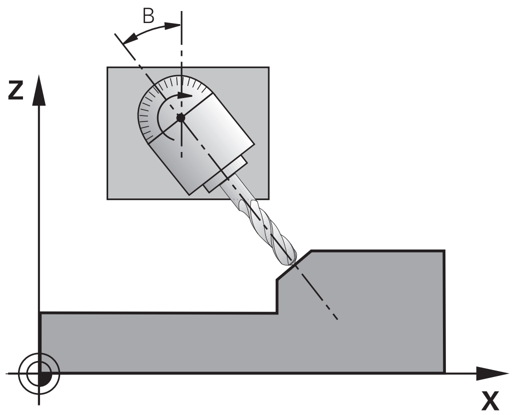
```
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC
   SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15
   CCY+20
28 CALL LBL 1
```

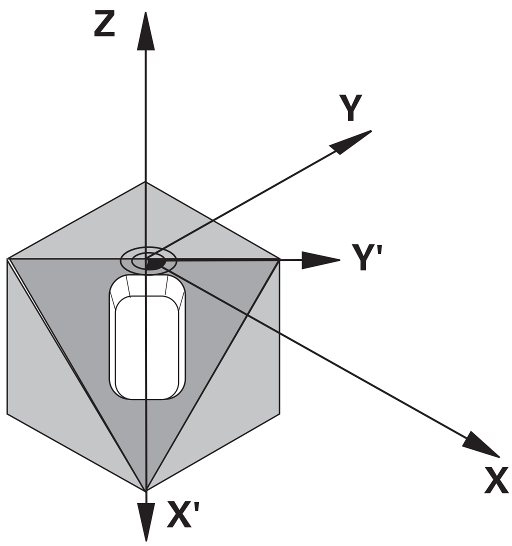
10.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, software option 1)

Effect

In Cycle 19 you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to 3 rotations (spatial angle) of the **fixed machine** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.





> Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

If you program the position of the working plane via spatial angles, the TNC will calculate the required angle positions of the tilted axes automatically and will store these in the parameters Q120 (A axis) to Q122 (C axis). If two solutions are possible, the TNC will choose the shorter path from the zero position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You must move all axes to activate compensation for all axes.

If you set the function **Tilting program run** to **Active** in the Manual Operation mode, the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.

Please note while programming:

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.

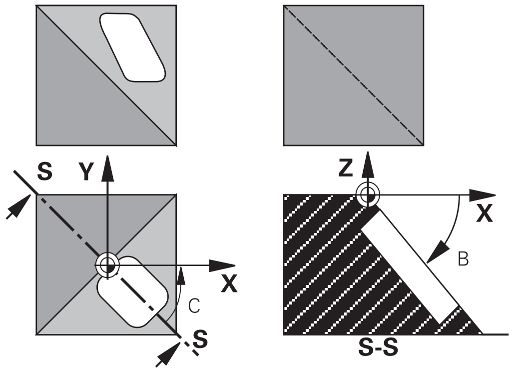
Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

The working plane is always tilted around the active datum.

If you use Cycle 19 when M120 is active, the TNC automatically rescinds the radius compensation, which also rescinds the M120 function.

Cycle parameters

- **Rotary axis and tilt angle?**: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range -360.000 to 360.000



If the TNC automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate? F=**: Traverse speed of the rotary axis during automatic positioning. Input range: 0 to 99999.999
- **Set-up clearance?** (incremental value): The TNC positions the tilting head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range: 0 to 99999.9999

Resetting

To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle once again and respond to the dialog question with the **NO ENT** key to disable the function.

Positioning the axes of rotation

The machine tool builder determines whether Cycle 19 positions the axes of rotation automatically or whether they must be positioned manually in the program. Refer to your machine manual.

Manual positioning of rotary axes

If the rotary axes are not positioned automatically in Cycle 19, you must position them in a separate L block after the cycle definition.

If you use axis angles, you can define the axis values right in the L block. If you use spatial angles, then use the Q parameters **Q120** (A-axis value), **Q121** (B-axis value) and **Q122** (C-axis value), which are described by Cycle 19.

For manual positioning, always use the rotary axis positions stored in Q parameters Q120 to Q122.

Avoid using functions, such as M94 (modulo rotary axes), in order to avoid discrepancies between the actual and nominal positions of rotary axes in multiple definitions.

Example NC blocks:

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the spatial angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0	
14 L A+Q120 C+Q122 R0 F1000	Position the rotary axes by using values calculated by Cycle 19
15 L Z+80 R0 FMAX	Activate compensation for the spindle axis
16 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle 19:

- The TNC can position only controlled axes
- In order for the tilted axes to be positioned, you must enter a feed rate and a set-up clearance in addition to the tilting angles, during cycle definition.
- Use only preset tools (the full tool length must be defined).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting
- The TNC performs the tilt at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

Example NC blocks:

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0 F5000 SETUP50	Also define the feed rate and the clearance
14 L Z+80 R0 FMAX	Activate compensation for the spindle axis
15 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Position display in the tilted system

On activation of Cycle 19, the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If necessary, the TNC outputs an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system.

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92) can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

1. Activate the datum shift
2. Activate tilting function
3. Activate rotation

...
Workpiece machining
...

1. Reset the rotation
2. Reset the tilting function
3. Reset the datum shift

Procedure for working with Cycle 19 WORKING PLANE

1 Write the program

- Define the tool (not required if TOOL.T is active), and enter the full tool length.
- Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece or clamping devices during tilting.
- If required, position the rotary axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes
- Traverse all principal axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- If required, define Cycle 19 WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle 19. You can define the new angular values directly.
- Reset Cycle 19 WORKING PLANE; program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle 19 and answer the dialog question with **NO ENT.**
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

2 Clamp the workpiece

3 Datum setting

- Manually by touch-off
- Controlled with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles User's Manual, chapter 2).
- Automatically with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles User's Manual, chapter 3).

4 Start the part program in the operating mode Program Run, Full Sequence

5 Manual Operation mode

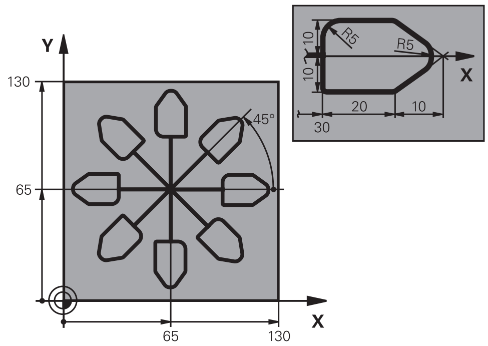
Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each rotary axis in the menu.

10.10 Programming Examples

Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- Machining within a subprogram



0 BEGIN PGM COTRANS MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+130 Y+130 Z+0	
3 TOOL CALL 1 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
6 CYCL DEF 7.1 X+65	
7 CYCL DEF 7.2 Y+65	
8 CALL LBL 1	Call milling operation
9 LBL 10	Set label for program section repeat
10 CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
11 CYCL DEF 10.1 IROT+45	
12 CALL LBL 1	Call milling operation
13 CALL LBL 10 REP 6/6	Return jump to LBL 10; repeat the milling operation six times
14 CYCL DEF 10.0 ROTATION	Reset the rotation
15 CYCL DEF 10.1 ROT+0	
16 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
17 CYCL DEF 7.1 X+0	
18 CYCL DEF 7.2 Y+0	
19 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
20 LBL 1	Subprogram 1
21 L X+0 Y+0 R0 FMAX	Define milling operation
22 L Z+2 R0 FMAX M3	
23 L Z-5 R0 F200	
24 L X+30 RL	
25 L IY+10	
26 RND R5	
27 L IX+20	
28 L IX+10 IY-10	

29 RND R5	
30 L IX-10 IY-10	
31 L IX-20	
32 L IY+10	
33 L X+0 Y+0 R0 F5000	
34 L Z+20 R0 FMAX	
35 LBL 0	
36 END PGM COTRANS MM	

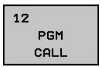
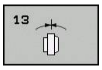
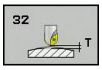
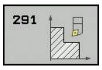
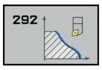
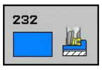
11

Cycles: Special Functions

11.1 Fundamentals

Overview

The TNC provides the following cycles for the following special purposes:

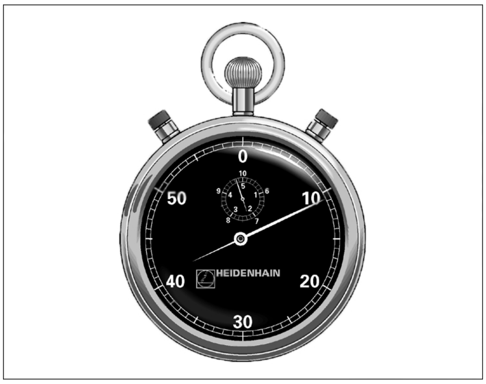
Cycle	Soft key	Page
9 DWELL TIME		279
12 PROGRAM CALL		280
13 SPINDLE ORIENTATION		282
32 TOLERANCE		283
225 ENGRAVING of texts		300
291 COUPLING TURNING INTERPOLATION		295
292 CONTOUR TURNING INTERPOLATION		286
232 FACE MILLING		304
239 ASCERTAIN THE LOAD		309

11.2 DWELL TIME (Cycle 9, DIN/ISO: G04)

Function

This causes the execution of the next block within a running program to be delayed by the programmed DWELL TIME. A dwell time can be used for such purposes as chip breaking.

The cycle becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



NC blocks

```
89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5
```

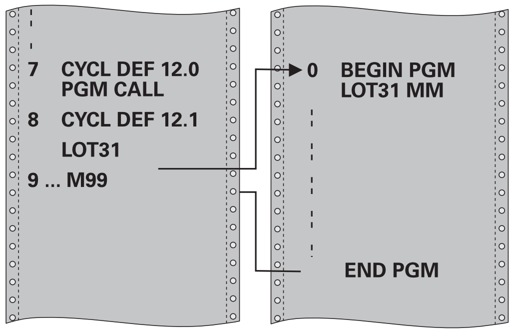
Cycle parameters

- **Dwell time in seconds**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

11.3 PROGRAM CALL (Cycle 12, DIN/ISO: G39)

Cycle function

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs. These can then be called like fixed cycles.



Please note while programming:

The program you are calling must be stored in the internal memory of your TNC.

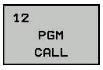
If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

If you want to define a DIN/ISO program to be a cycle, enter the file type .I behind the program name.

As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called program can also influence the calling program.

Cycle parameters



- **Program name**: Enter the name of the program you want to call and, if necessary, the directory it is located in or
- Activate the file select dialog with the **SELECT** soft key and select the program to be called

Call the program with:

- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Designate program 50 as a cycle and call it with M99

```
55 CYCL DEF 12.0 PGM CALL
56 CYCL DEF 12.1 PGM TNC:
    \KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99
```

11.4 SPINDLE ORIENTATION (Cycle 13, DIN/ISO: G36)

Cycle function

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

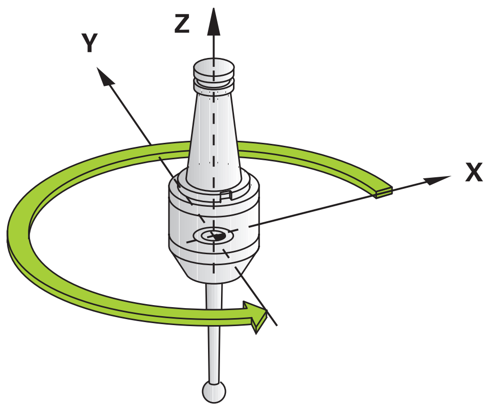
The TNC can control the machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set by the machine manufacturer (see your machine manual).



NC blocks

```
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180
```

Please note while programming:

Cycle 13 is used internally for Cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

- **Angle of orientation**: Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°

11.5 TOLERANCE (Cycle 32, DIN/ISO: G62)

Cycle function

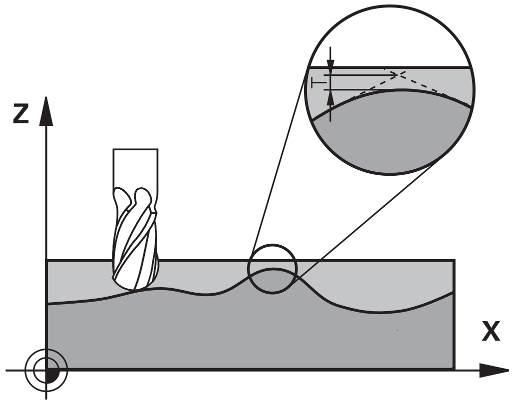
Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

With the entries in Cycle 32 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the TNC has been adapted to the machine's characteristics.

The TNC automatically smoothens the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without short pauses for computing time. **Even if the TNC does not move with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the TNC can move the axes.

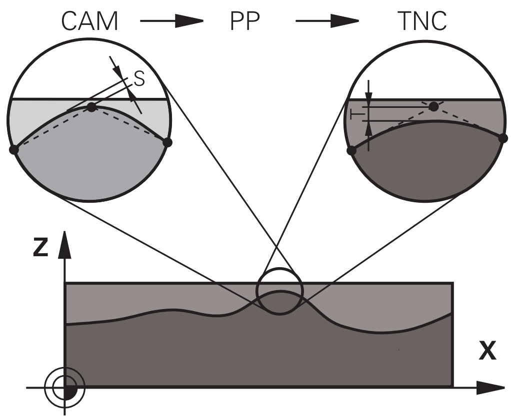
Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**tolerance value**) is set in a machine parameter by the machine manufacturer. With **CYCLE 32** you can change the pre-set tolerance value and select different filter settings, provided that your machine tool builder has implemented these features.



Influences of the geometry definition in the CAM system

The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system.
The maximum point spacing of NC programs generated in a postprocessor (PP) is defined through the chord error. If the chord error is less than or equal to the tolerance value **T** defined in Cycle 32, then the TNC can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing if in Cycle 32 you choose a tolerance value between 110-% and 200-% of the CAM chord error.



Please note while programming:

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the TNC, but by the fact that, in order to machine the contour element transitions very exactly, the TNC might have to drastically reduce the speed.

Cycle 32 is DEF active which means that it becomes effective as soon as it is defined in the part program.

The TNC resets Cycle 32 if you

- Redefine it and confirm the dialog question for the **tolerance value** with **NO ENT**.
- Select a new program with the **PGM MGT** key.

After you have reset Cycle 32, the TNC reactivates the tolerance that was predefined by machine parameter.

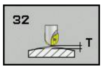
In a program with millimeters set as unit of measure, the TNC interprets the entered tolerance value in millimeters. In an inch program it interprets it as inches.

If you transfer a program with Cycle 32 that contains only the cycle parameter **Tolerance value T**, the TNC inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases, except if HSC filters are active on your machine (settings made by the machine tool builder).

If Cycle 32 is active, the TNC shows the parameters defined for Cycle 32 on the **CYC** tab of the additional status display.

Cycle parameters



- **Tolerance value T**: Permissible contour deviation in mm (or inches with inch programming). Input range 0 to 99999.9999
- **HSC MODE, Finishing=0, Roughing=1**: Activate filter:
  - Input value 0: **Milling with increased contour accuracy**. The TNC uses internally defined finishing filter settings
  - Input value 1: **Milling at an increased feed rate**. The TNC uses internally defined roughing filter settings
- **Tolerance for rotary axes TA**: Permissible position error of rotary axes in degrees when M128 is active (FUNCTION TCPM). The TNC always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for programs for more than one axis by entering a large tolerance value (e.g. 10°), since the TNC does not always have to move the rotary axis to the given nominal position. The contour will not be damaged by entering a rotary axis tolerance value. Only the position of the rotary axis with respect to the workpiece surface will change. Input range 0 to 179.9999

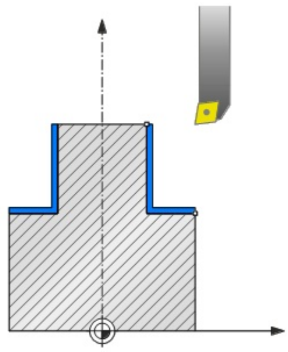
NC blocks

```
95 CYCL DEF 32.0 TOLERANCE
96 CYCL DEF 32.1 T0.05
97 CYCL DEF 32.2 HSC-MODE:1 TA5
```

11.6 CONTOUR TURNING INTERPOLATION (Cycle 292, DIN/ISO: G292, software option 96)

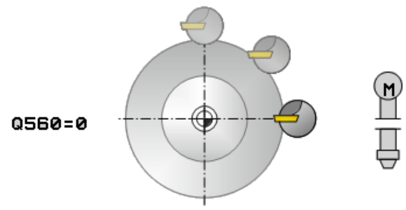
Cycle run

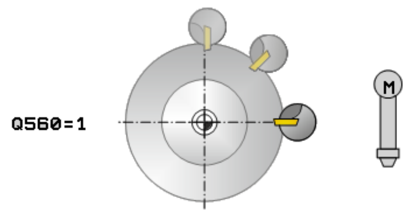
Cycle 292 CONTOUR TURNING INTERPOLATION couples the tool spindle to the position of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. Cycle 292 CONTOUR TURNING INTERPOLATION is run in milling mode and is CALL-active. After executing this cycle, the TNC deactivates the spindle coupling again.



Before using Cycle 292, you first need to define the desired contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. By entering Q560=1 you can turn the contour; a cutting edge of the tool is oriented to the center of a circle. When you enter Q560=0 you can mill the contour; the spindle is not oriented in this case.

Cycle run, Q560=1: Contour turning





1 The TNC first stops the spindle (M5).
2 The TNC orients the tool spindle to the specified center of rotation. The specified angle Q336 is taken into account. If defined, the "ORI" value from the turning tool table (toolturn.trn) is also considered.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 The TNC positions the tool to the contour start radius Q491, taking the selected machining operation inside/outside Q529 and the clearance to side Q357 into account. The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**
5 The TNC uses interpolation turning to machine the defined contour. In interpolation turning the linear axes of the working plane move on a circle, whereas the spindle axis is oriented perpendicularly to the surface.
6 At the end point of the contour, the TNC retracts the tool perpendicularly by the set-up clearance.
7 Finally, the TNC retracts the tool to the clearance height.
8 The TNC now automatically deactivates the coupling of the tool spindle to the linear axes.

Cycle run, Q560=0: Contour milling

1 The M3/M4 function programmed before the cycle call remains in effect.
2 No spindle stop and **no** spindle orientation are performed. Q336 is not taken into account.
3 The TNC positions the tool to the contour start radius Q491, taking the selected machining operation inside/outside Q529 and the clearance to side Q357 into account. The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**
4 The TNC machines the defined contour with the spindle rotating (M3/M4). In the process, the reference axes of the working plane move on a circle; the TNC does not orient the tool spindle.
5 At the end point of the contour, the TNC retracts the tool perpendicularly by the set-up clearance.
6 Finally, the TNC retracts the tool to the clearance height.

Please note while programming:

A programming example is provided at the end of this chapter, see page 313.

Program the contour either with monotonically decreasing or monotonically increasing coordinates.

When programming, remember to use only positive radius values.

Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

Roughing operations with multiple passes are not possible in this cycle.

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

For inside machining, the TNC checks whether the active tool radius is smaller than half the contour start diameter Q491 plus the clearance to side Q357. If the check shows that the tool is too large, the program is aborted.

If Cycle 8 MIRRORING is active, the TNC does **not** execute the interpolation turning cycle.

If Cycle 26 SCALING FACTOR is active and the scaling factor in an axis is not equal to 1, the TNC does **not** execute the interpolation turning cycle.

The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

The center of the turning contour is the starting point in the working plane at the time the cycle is called.

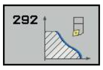
This cycle is effective only for machines with servo-controlled spindle.

Software option 96 must be enabled.

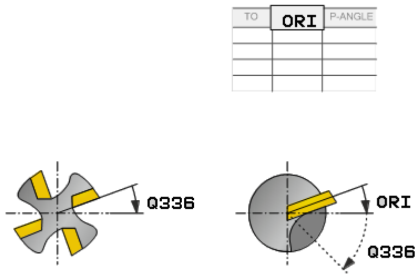
If Q560=1 the TNC does not check whether the spindle is rotating when the cycle is run. (Independent of CfgGeoCycle – displaySpindleError)

Your TNC might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

Cycle parameters



- **Spindle coupling (0, 1)** Q560: Define whether the spindle is to be coupled.
  0: Spindle coupling off (contour milling)
  1: Spindle coupling on (contour turning)
- **Angle of spindle** Q336: The TNC orients the tool to this angle before starting the machining operation. If you are using a milling cutter, orient a cutting edge to the center of rotation. If you have defined the "ORI" value in the tool table, this value is also taken into account for spindle orientation. Input range 0.000 to 360.000
- **Change tool direction (3, 4)** Q546: Direction of spindle rotation of the active tool:
  3: Right-turning tool (M3)
  4: Left-turning tool (M4)
- **Machining operation (+1, 0)** Q529: Define whether to perform inside or outside machining:
  +1: Inside machining
  0: Outside machining
- **Surface oversize** Q221: Allowance in the working plane. Input range 0 to 99.9999
- **Infeed** Q441 (mm/rev): Amount that the tool advances with each revolution. Input range 0.001 to 99.999
- **Feed rate** Q449 (mm/min): Feed rate with respect to the contour starting point Q491. Input range 0.1 to 99999.9. The feed rate of the tool's center point path is adjusted according to the tool radius and the machining operation Q529. From these parameters, the TNC determines the programmed cutting speed at the diameter of the contour starting point.
  Q529=1: The feed rate of the tool center point path is reduced for inside machining
  Q529=0: The feed rate of the tool center point path is increased for outside machining
- **Contour start radius** Q491 (absolute value): Radius of the contour starting point (e.g. X coordinate, with tool axis Z). Input range 0.9999 to 99999.9999
- **Clearance to side** Q357 (incremental): Clearance between the side of the tool and the workpiece when approaching for the first plunging depth Input range 0 to 99999.9
- **Clearance height** Q445 (absolute): Absolute height at which the tool cannot collide with the workpiece. The tool retracts to this position at the end of the cycle. Input range -99999.9999 to 99999.9999



NC blocks

```
63 CYCL DEF 292 CONTOUR
  TURNING INTERPOLATION
  Q560=1     ;SPINDLE COUPLING
  Q336=0     ;ANGLE OF SPINDLE
  Q546=3     ;CHANGE TOOL
              DIRECTN.
  Q529=0     ;MACHINING
              OPERATION
  Q221=0     ;SURFACE OVERSIZE
  Q441=0.5   ;INFEED
  Q449=2000;FEED RATE
  Q491=0     ;CONTOUR START DIA.
  Q357=2     ;CLEARANCE TO SIDE
  Q445=50    ;CLEARANCE HEIGHT
```

Machining variants

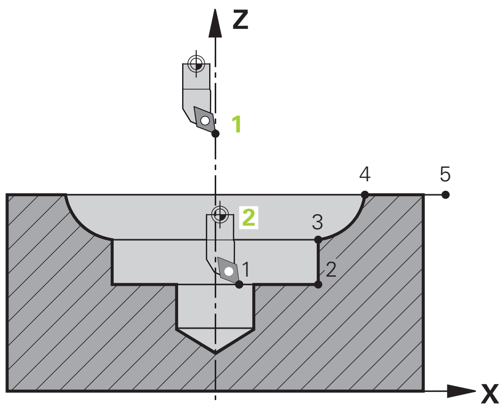
Before using Cycle 292, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

Tool axis used	Axial coordinate	Radial coordinate
Z	Z	X
X	X	Y
Y	Y	Z

Example: If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius of the contour in X.

You can use this cycle for inside and outside machining. The following information illustrates some of the notes given in the "Please note while programming" section. You can also find a programming example in "Example: Interpolation Turning Cycle 292", page 313

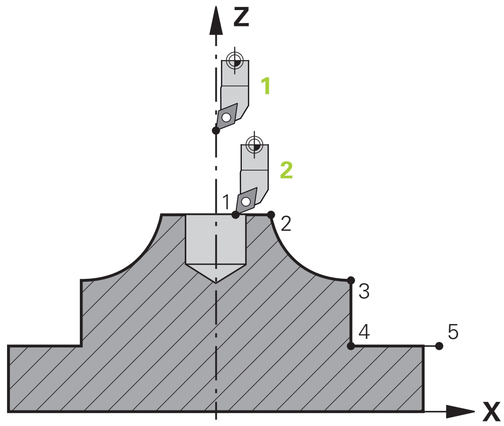
Inside machining



- The center of rotation is the position of the tool in the working plane at the time the cycle is called 1
- **After the cycle is started, neither the indexable insert nor the spindle center must be moved into the center of rotation!** Keep this in mind when describing the contour! 2
- The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

  When programming an inside contour, please also remember:
  - Program either monotonically increasing radial and axial coordinates, e.g. 1-5
  - Or program monotonically decreasing radial and axial coordinates, e.g. 5-1
  - Program inside contours with a radius greater than the tool radius.

Outside machining



- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **After the cycle is started, neither the indexable insert nor the spindle center must be moved into the center of rotation.** Keep this in mind when describing the contour! **2**
- The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

  When programming an outside contour, please also remember:
  - Program either monotonically increasing radial and monotonically decreasing axial coordinates, e.g. 1-5
  - Or program monotonically decreasing radial and monotonically increasing axial coordinates, e.g. 5-1
  - Program outside contours with a radius greater than 0.

Defining the tool

Overview

Depending on the setting of the parameter Q560, you can mill (Q560=0) or turn (Q560=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define the turning tool in the tool table (tool.t) as a milling cutter. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). Orient the turning tool to the spindle center and enter this spindle orientation angle in the parameter Q336 of the cycle. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

  > The tool holder is not monitored! If the rotation diameter resulting from the tool holder is greater than that from the cutting edge, the machine operator must take this into account for inside machining.

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can use a milling cutter for interpolation turning. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). Orient a cutting edge of the milling cutter to the spindle center and enter this angle in the parameter Q336. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define the turning tool in the turning tool table (toolturn.trn). In this case, the spindle is oriented to the center of rotation by taking tool-specific data into account, such as the machining operation (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and the parameter Q336.

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + Q336
Interpolation turning, inside	7	ORI + Q336 + 180
Interpolation turning, outside	7	ORI + Q336 + 180
Interpolation turning, inside	1	ORI + Q336
Interpolation turning, outside	8,9	ORI + Q336
Interpolation turning, inside	8,9	ORI + Q336

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1 or 7
- TYPE: FINISH, with the machining directions TO: 1 or 7
- TYPE: BUTTON, with the machining directions TO: 1 or 7

For inside machining, the TNC checks whether the active tool radius is smaller than half the contour start diameter Q491 plus the clearance to side Q357. If the check shows that the tool is too large, the program is aborted.

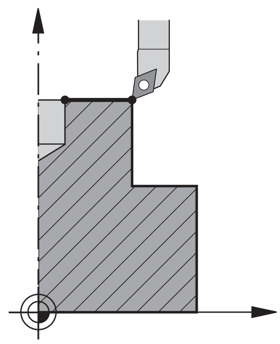
The following tool types cannot be used for interpolation turning: (error message "Function not possible with this tool type" is displayed)

- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

11.7 COUPLING TURNING INTERPOLATION (Cycle 291, DIN/ISO: G291, software option 96)

Cycle run

Cycle 291 COUPLING TURNING INTERPOLATION couples the tool spindle to the position of the linear axes or deactivates this spindle coupling. In interpolation turning the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates Q216 and Q217. Cycle 291 COUPLING TURNING INTERPOLATION is run in milling mode and is CALL-active.



Cycle run if Q560=1:

1 The TNC first stops the spindle (M5).
2 The TNC orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation Q336 is taken into account. If an "ORI" value is given in the tool table, it is also taken into account if so defined.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 To terminate the cycle, the coupling must be deactivated by the operator. (Through Cycle 291 or a new program selection)

Cycle run if Q560=0:

1 The TNC deactivates the spindle coupling.
2 The tool spindle is no longer coupled to the position of the linear axes.
3 Machining with the interpolation turning cycle 291 is terminated.
4 If Q560=0, the parameters Q336, Q216, Q217 are irrelevant.

Please note while programming:

After defining Cycle 291, program the desired machining operation, e.g. by using linear/polar blocks. In addition to programming the paths, you also program the rotation of the tool. As the position is coupled to the linear axes, you do not have to activate spindle rotation—programming M3/M4 thus can be omitted. A programming example is provided at the end of this chapter, see page 311.

Cycle 291 is CALL-active.

If you define the turning tool in the turning tool table (toolturn.trn), do not enter a tool radius compensation in the contour description.

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

This cycle can also be used in a tilted working plane.

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

If Cycle 8 MIRRORING is active, the TNC does **not** execute the interpolation turning cycle.

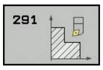
If Cycle 26 SCALING FACTOR is active and the scaling factor in an axis is not equal to 1, the TNC does **not** execute the interpolation turning cycle.

This cycle is effective only for machines with servo-controlled spindle.

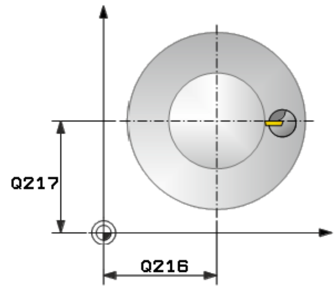
Your TNC might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

Software option 96 must be enabled.

Cycle parameters



- **Spindle coupling (0, 1)** Q560: Define whether the tool spindle is coupled to the position of the linear axes. When spindle coupling is active, a cutting edge of the tool is oriented to the center of rotation.
  0: Spindle coupling off
  1: Spindle coupling on
- **Angle of spindle** Q336: The TNC orients the tool to this angle before starting the machining operation. If you are using a milling cutter, orient a cutting edge to the center of rotation. If you have defined the "ORI" value in the tool table, this value is also taken into account for spindle orientation. Input range 0.000 to 360.000
- **Center in 1st axis** Q216 (absolute): Center of rotation in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q217 (absolute): Center of rotation in the minor axis of the working plane. Input range -99999.9999 to 99999.9999



NC blocks

64 CYCL DEF 291 COUPLING TURNING INTERPOLATION	
Q560=1	;SPINDLE COUPLING
Q336=0	;ANGLE OF SPINDLE
Q216=50	;CENTER IN 1ST AXIS
Q217=50	;CENTER IN 2ND AXIS

Defining the tool

Overview

Depending on the setting of the parameter Q560, you can activate (Q560=1) or deactivate (Q560=0) the COUPLING TURNING INTERPOLATION cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

> Q560=0: Deactivate the **COUPLING TURNING INTERPOLATION** cycle!

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter Q560 to 1, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define the turning tool in the tool table (tool.t) as a milling cutter. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Orient the turning tool to the spindle center and enter this spindle orientation angle in the parameter Q336 of the cycle. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

  > The tool holder is not monitored! If the rotation diameter resulting from the tool holder is greater than that from the cutting edge, the machine operator must take this into account for inside machining.

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can use a milling cutter for interpolation turning. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). Orient a cutting edge of the milling cutter to the spindle center and enter this angle in the parameter Q336. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

- **Define a turning tool in the turning tool table (toolturn.trn)**
  If you are working with option 50, you can define the turning tool in the turning tool table (toolturn.trn). In this case, the spindle is oriented to the center of rotation by taking tool-specific data into account, such as the machining operation (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and the parameter Q336.

> If you define the turning tool in the turning tool table (toolturn.trn), do not enter a tool radius compensation in the contour description.

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + Q336
Interpolation turning, inside	7	ORI + Q336 + 180
Interpolation turning, outside	7	ORI + Q336 + 180
Interpolation turning, inside	1	ORI + Q336
Interpolation turning, outside	8,9	ORI + Q336
Interpolation turning, inside	8,9	ORI + Q336

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1 or 7
- TYPE: FINISH, with the machining directions TO: 1 or 7
- TYPE: BUTTON, with the machining directions TO: 1 or 7

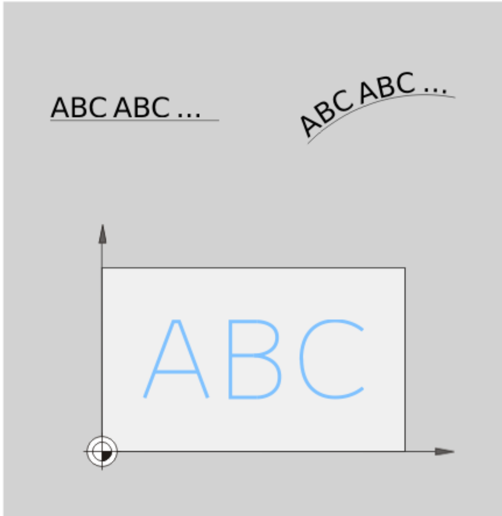
> **The following tool types cannot be used for interpolation turning:** (error message "Function not possible with this tool type" is displayed)
> - TYPE: ROUGH, with the machining directions TO: 2 to 6
> - TYPE: FINISH, with the machining directions TO: 2 to 6
> - TYPE: BUTTON, with the machining directions TO: 2 to 6
> - TYPE: RECESS
> - TYPE: RECTURN
> - TYPE: THREAD

11.8 ENGRAVING (Cycle 225, DIN/ISO: G225)

Cycle run

This cycle is used to engrave texts on a flat surface of the workpiece. The texts can be arranged in a straight line or along an arc.

1 The TNC positions the tool in the working plane to the starting point of the first character.
2 The tool plunges perpendicularly to the engraving floor and mills the character. The TNC retracts the tool to the set-up clearance between the characters when required. After machining the character, the tool is at the set-up clearance above the workpiece surface.
3 This process is repeated for all characters to be engraved.
4 Finally, the TNC retracts the tool to the 2nd set-up clearance.



Please note while programming:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

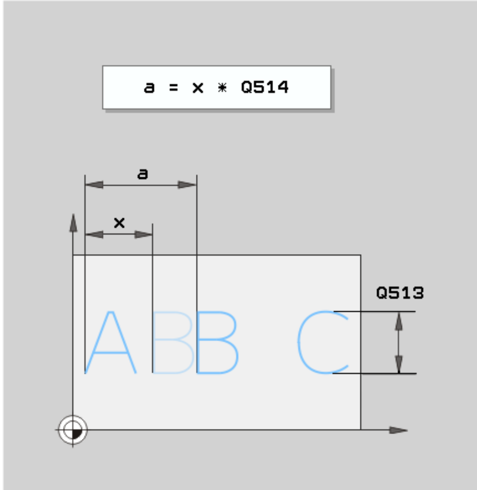
If you engrave the text in a straight line (**Q516=0**), the starting point of the first character is determined by the tool position at the time the cycle is called.

If you engrave the text along an arc (**Q516=1**), the arc's center is determined by the tool position at the time the cycle is called.

The text to be engraved can also be transferred with a string variable (**QS**).

Cycle parameters

- **Engraving text** QS500: Text to be engraved inside quotation marks. Assignment of a string variable through the Q key of the numerical keypad. The Q key on the ASCI keyboard represents normal text input. Allowed entry characters: see "Engraving system variables", page 303
- **Character height** Q513 (absolute): Height of the characters to be engraved in mm. Input range 0 to 99999.9999
- **Space factor** Q514: The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the TNC if you program Q514 = 0. If Q514 is not equal to 0, the TNC scales the space between the characters. Input range 0 to 9.9999
- **Font** Q515: Currently without function
- **Text in a straight line/on arc (0/1)** Q516: Engrave text in a straight line: Input = 0 Engrave text on an arc: Input = 1
- **Angle of rotation** Q374: Center angle if the text is to be arranged on an arc. Engraving angle if the text is arranged in a straight line. Input range -360.0000 to +360.0000°
- **Radius of text on an arc** Q517 (absolute): Radius of the arc in mm on which the TNC is to arrange the text. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Depth** Q201 (incremental value): Distance between workpiece surface and engraving floor
- **Feed rate for plunging** Q206: Traversing speed of the tool when moving into the workpiece in mm/min. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**



NC blocks

62 CYCL DEF 225 ENGRAVING	
QS500="A"	**;ENGRAVING TEXT**
Q513=10	**;CHARACTER HEIGHT**
Q514=0	**;SPACE FACTOR**
Q515=0	**;FONT**
Q516=0	**;TEXT LAYOUT**
Q374=0	**;ANGLE OF ROTATION**
Q517=0	**;CIRCLE RADIUS**
Q207=750	**;FEED RATE FOR MILLING**
Q201=-0.5	**;DEPTH**
Q206=150	**;FEED RATE FOR PLNGNG**
Q200=2	**;SET-UP CLEARANCE**
Q203=+20	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers:

! # $ % & ' ( ) * + , - . / : ; < = > ? @ [\ ] _ ß CE

> The TNC uses the special characters **%** and **** for special functions. These characters must be indicated twice in the text to be engraved (e.g. **%%**) if you want to engrave them.

When engraving German umlauts, ß, ø, @ or the CE character, enter the character % before the character to be engraved:

Algebraic sign	Input
ä	%ae
ö	%oe
ü	%ue
Ä	%AE
Ö	%OE
Ü	%UE
ß	%ss
ø	%D
@	%at
CE	%CE

Characters that cannot be printed

Apart from text, you can also define certain non-printable characters for formatting purposes. Enter the special character **** before the non-printable characters.

The following formatting possibilities are available:

Character	Input
Line break	\n
Horizontal tab (the tab width is permanently set to 8 characters)	\t
Vertical tab (the tab width is permanently set to one line)	\v

Engraving system variables

In addition to the standard characters, you can engrave the contents of certain system variables. Enter **%** before the system variable.

You can also engrave the current date or time. Enter **%time<x>**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY. (Identical to the function **SYSSTR ID332**, see the User's Manual for Conversational Programming, "Q parameter programming" chapter, "Copying system data to a string" section)

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **time08**.

Character	Input
DD.MM.YYYY hh:mm:ss	%time00
D.MM.YYYY h:mm:ss	%time01
D.MM.YYYY h:mm	%time02
D.MM.YY h:mm	%time03
YYYY-MM-DD hh:mm:ss	%time04
YYYY-MM-DD hh:mm	%time05
YYYY-MM-DD h:mm	%time06
YY-MM-DD h:mm	%time07
DD.MM.YYYY	%time08
D.MM.YYYY	%time09
D.MM.YY	%time10
YYYY-MM-DD	%time11
YY-MM-DD	%time12
hh:mm:ss	%time13
h:mm:ss	%time14
h:mm	%time15

11.9 FACE MILLING (Cycle 232, DIN/ISO: G232)

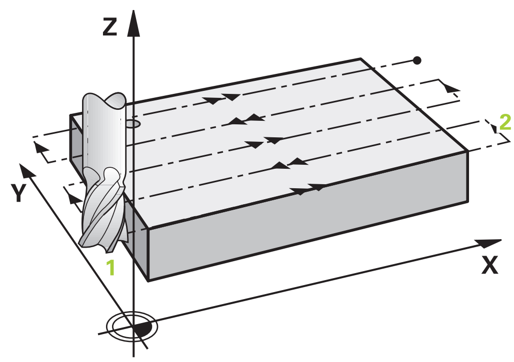
Cycle run

Cycle 232 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

1 From the current position, the TNC positions the tool at rapid traverse **FMAX** to the starting position using positioning logic **1**: If the current position in the spindle axis is greater than the 2nd set-up clearance, the control positions the tool first in the machining plane and then in the spindle axis. Otherwise it first moves to the 2nd set-up clearance and then in the machining plane. The starting point in the machining plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2 The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0



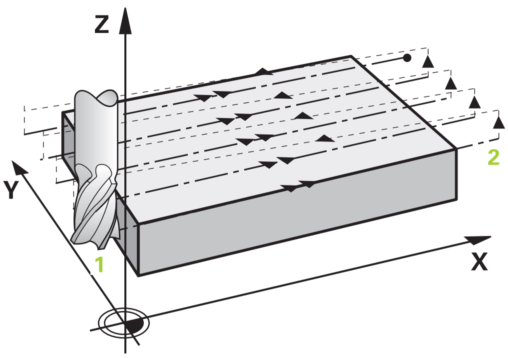
3 The tool subsequently advances to the stopping point **2** at the programmed feed rate for milling. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted in **FMAX** to the 2nd set-up clearance.

Strategy Q389=1

3 The tool subsequently advances to the end point **2** at the programmed feed rate for milling. The end point lies **at the edge** of the surface. The TNC calculates the end point from the programmed starting point, the programmed length and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**. The motion to the next line again occurs at the edge of the workpiece.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted in **FMAX** to the 2nd set-up clearance.

Strategy Q389=2

3 The tool subsequently advances to the stopping point **2** at the programmed feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.
4 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at the pre-positioning feed rate directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then returns to the current infeed depth and moves in the direction of the next end point **2**.
6 The multipass process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted in **FMAX** to the 2nd set-up clearance.

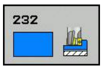


Please note while programming:

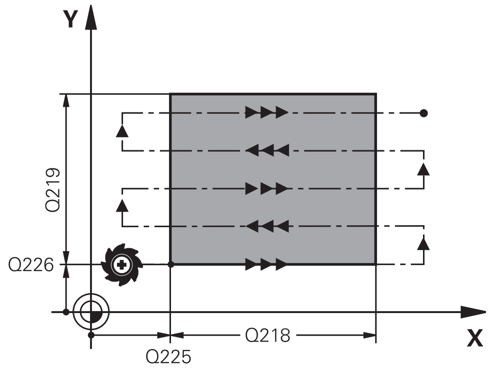
Enter the **2nd set-up clearance** in Q204 so that no collision with the workpiece or the fixtures can occur.

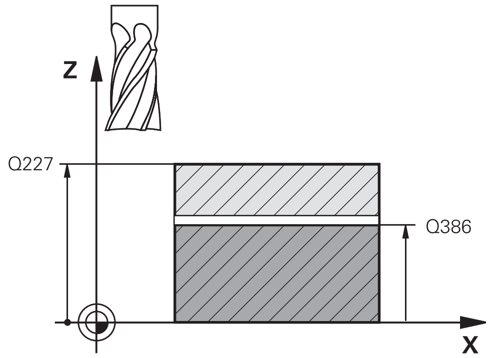
If the starting point in the 3rd axis Q227 and the end point in the 3rd axis Q386 are entered as equal values, the TNC does not run the cycle (depth = 0 has been programmed).

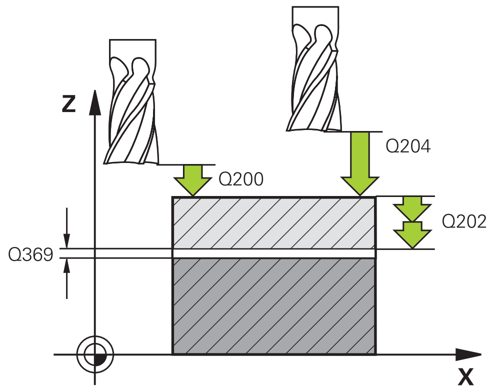
Cycle parameters



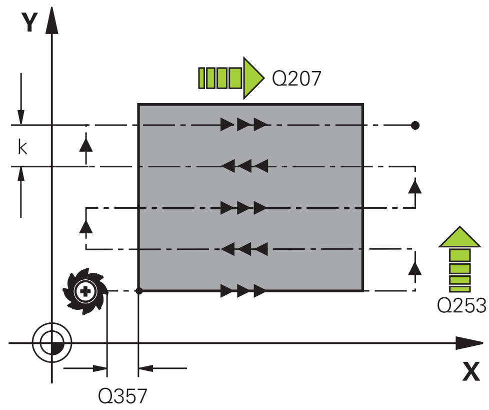
- **Machining strategy (0/1/2)** Q389: Determine how the TNC should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
- **Starting point in 1st axis** Q225 (absolute): Starting point coordinate of the surface to be machined in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 2nd axis** Q226 (absolute): Starting point coordinate of the surface to be machined in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 3rd axis** Q227 (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999
- **End point in 3rd axis** Q386 (absolute): Coordinate in the spindle axis to which the surface is to be face milled. Input range -99999.9999 to 99999.9999
- **1st side length** Q218 (incremental value): Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path in reference to the **starting point in the 1st axis**. Input range -99999.9999 to 99999.9999
- **2nd side length** Q219 (incremental value): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **starting point in the 2nd axis**. Input range -99999.9999 to 99999.9999
- **Maximum plunging depth** Q202 (incremental value): **Maximum** amount that the tool is advanced each time. The TNC calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range 0 to 99999.9999
- **Allowance for floor** Q369 (incremental): Distance used for the last infeed. Input range 0 to 99999.9999







- **Max. path overlap factor** Q370: **Maximum** stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. tooth radius when using a face-milling cutter), the TNC reduces the stepover accordingly. Input range 0.1 to 1.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min, while milling the last infeed. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207. Input range 0 to 99999.9999, alternatively **FMAX**, **FAUTO**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the TNC moves the tool at the set-up clearance over the current plunging depth to the starting point of the next pass. Input range 0 to 99999.9999
- **Clearance to side** Q357 (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used. Input range 0 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**



NC blocks

71 CYCL DEF 232 FACE MILLING	
Q389=2	;STRATEGY
Q225=+10	;STARTNG PNT 1ST AXIS
Q226=+12	;STARTNG PNT 2ND AXIS
Q227=+2.5	;STARTNG PNT 3RD AXIS
Q386=-3	;END POINT 3RD AXIS
Q218=150	;FIRST SIDE LENGTH
Q219=75	;2ND SIDE LENGTH
Q202=2	;MAX. PLUNGING DEPTH
Q369=0.5	;ALLOWANCE FOR FLOOR
Q370=1	;MAX. TOOL PATH OVERLAP
Q207=500	;FEED RATE FOR MILLING
Q385=800	;FINISHING FEED RATE
Q253=2000	;F PRE-POSITIONING
Q200=2	;SET-UP CLEARANCE
Q357=2	;CLEARANCE TO SIDE
Q204=2	;2ND SET-UP CLEARANCE

11.10 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143)

Cycle run

The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of table axes. Option 143 LAC (Load Adaptive Control) and Cycle 239 ASCERTAIN THE LOAD enable the control to automatically ascertain and adapt the current mass moment of inertia of the load as well as the current friction forces, or to reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The TNC performs a weighing procedure to ascertain the weight acting on the axes. In the weighing procedure, the axes move a specified distance—the machine tool builder defines the exact scope of axis movement. Before weighing, the axes are moved to a position, if required, where there is no danger of collision during the weighing procedure. This safe position is defined by the machine tool builder.

Parameter Q570 = 0

1 There is no physical movement of the axes.
2 The TNC resets the LAC.
3 The TNC activates feedforward parameters and, if applicable, controller parameters that ensure a safe movement of the axes concerned, regardless of the load condition—the parameters set with Q570=0 are **independent** of the current load.
4 These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

1 The TNC performs a weighing procedure in which it moves one or more axes. Which axes are moved depends on the setup of the machine and on the drives of the axes.
2 The scope of axis movement is defined by the machine tool builder.
3 The feedforward and controller parameters determined by the TNC **depend** on the current load.
4 The TNC activates the parameters determined.

Please note while programming:

Cycle 239 becomes effective immediately after definition.

If you are using the mid-program startup function and the TNC skips Cycle 239 in the block scan, the TNC will ignore this cycle—no weighing procedure will be performed.

The machine must be prepared by the machine tool builder for this cycle.

Cycle 239 can only be used with option 143 LAC (Load Adaptive Control).

This cycle may lead to extensive motion in one or more axes!

The TNC moves the axes at rapid traverse.

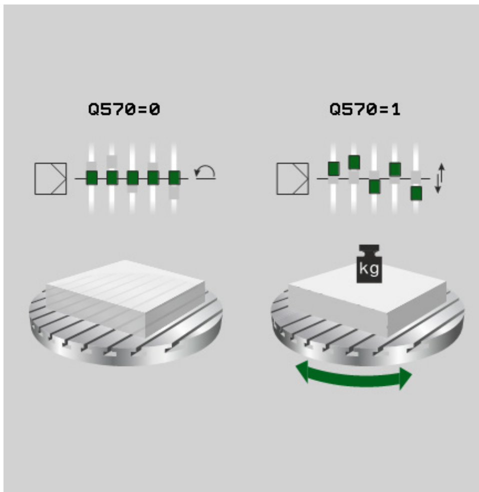
Set the potentiometer for feed-rate and rapid-traverse override to at least 50 % to ensure a correct ascertainment of the load.

Before cycle start, the TNC might move to a safe position; this position is defined by the machine tool builder!

Before using this cycle, contact your machine tool builder for details on the type and scope of movements performed in Cycle 239!

Cycle parameters

- **ASCERTAIN THE LOAD** Q570: Define whether the TNC is to perform the LAC (Load Adaptive Control) weighing procedure or reset the last determined load-dependent feedforward and controller parameters:
  0: Reset LAC; the values last set by the TNC are reset; the TNC uses load-independent feedforward and controller parameters
  1: Perform the weighing procedure; the TNC moves the axes to determine the feedforward and controller parameters with respect to the current load; the determined values are activated immediately



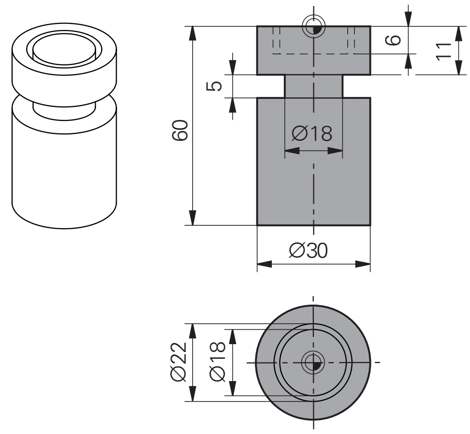
NC blocks

```
62 CYCL DEF 239 ASCERTAIN THE LOAD
   Q570=+0    ;ASCERTAIN THE LOAD
```

11.11 Programming examples

Example: Interpolation Turning Cycle 291

Cycle 291 COUPLING TURNING INTERPOLATION is used in the following program. This programming example illustrates the machining of an axial recess and a radial recess.



Program sequence

- Tool call: Recessing tool for axial recess
- Start of interpolation turning: Description and call of Cycle 291; Q560=1
- Machining the axial recess
- End of interpolation turning: Description and call of Cycle 291; Q560=0
- Tool call: Recessing tool for radial recess
- Start of interpolation turning: Description and call of Cycle 291; Q560=1
- Machining the radial recess
- End of interpolation turning: Description and call of Cycle 291; Q560=0

0 BEGIN PGM 1 MM	
1 BLK FORM CYLINDER Z R15 L60	Definition of workpiece blank: Cylinder
2 TOOL CALL 10 Z	Tool call: Recessing tool for axial recess
3 CC X+0 Y+0	
4 LP PR+30 PA+0 R0 FMAX	Retract the tool
5 CYCL DEF 291 COUPLG. TURNG. INTERP.	Activate interpolation turning
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
6 CYCL CALL	Call the cycle
7 LP PR+9 PA+0 RR FMAX	Position the tool in the working plane
8 L Z+10 FMAX	
9 L Z+0.2 F2000	Position the tool in the spindle axis
10 LBL 1	Recessing on face, infeed: 0.2 mm, depth: 6 mm
11 CP IPA+360 IZ-0.2 DR+ F10000	
12 CALL LBL 1 REP 30	
13 LBL 2	Retract from recess, step: 0.4 mm
14 CP IPA+360 IZ+0.4 DR+	
15 CALL LBL 2 REP15	
16 L Z+200 R0 FMAX	Retract to clearance height, deactivate radius compensation
17 CYCL DEF 291 COUPLG. TURNG. INTERP.	Terminate interpolation turning
Q560=+0 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	

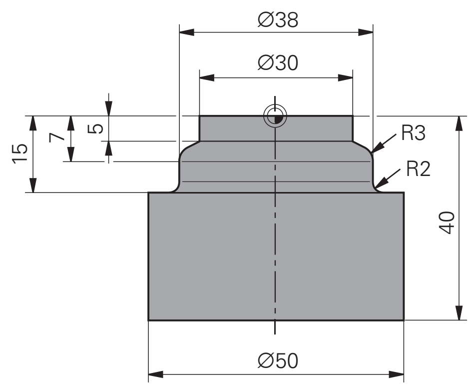
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
18 CYCL CALL	Call the cycle
19 TOOL CALL 11 Z	Tool call: Recessing tool for radial recess
20 CC X+0 Y+0	
21 LP PR+25 PA+0 R0 FMAX	Retract the tool
22 CYCL DEF 291 COUPLG. TURNG. INTERP.	Activate interpolation turning
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
23 CYCL CALL	Call the cycle
24 LP PR+15.2 PA+0 RR FMAX	Position the tool in the working plane
25 L Z+10 FMAX	
26 L Z-11 F7000	Position the tool in the spindle axis
27 LBL 3	Recessing on lateral surface, infeed: 0.2 mm, depth: 6 mm
28 CC X+0.1 Y+0	
29 CP IPA+180 DR+ F10000	
30 CC X-0.1 Y+0	
31 CP IPA+180 DR+	
32 CALL LBL 3 REP15	
33 LBL 4	Retract from recess, step: 0.4 mm
34 CC X-0.2 Y+0	
35 CP IPA+180 DR+	
36 CC X+0.2 Y+0	
37 CP IPA+180 DR+	
38 CALL LBL 4 REP8	
39 L LP PR+25 FMAX	
40 Z+200 R0 FMAX	Retract to clearance height, deactivate radius compensation
41 CYCL DEF 291 COUPLG. TURNG. INTERP.	Terminate interpolation turning
Q560=+0 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
42 CYCL CALL	Call the cycle
43 M30	
44 END PGM 1 MM	

Example: Interpolation Turning Cycle 292

Cycle 292 CONTOUR TURNING INTERPOLATION is used in the following program. This programming example illustrates the machining of an outside contour with the milling spindle rotating.

Program sequence

- Tool call: Milling cutter D20
- Cycle 32 Tolerance
- Reference to the contour with Cycle 14
- Cycle 292 Contour turning interpolation



0 BEGIN PGM 2 MM	
1 BLK FORM CYLINDER Z R25 L40	Definition of workpiece blank: Cylinder
2 TOOL CALL "D20" Z S111	Tool call: End mill D20
3 CYCL DEF 32.0 TOLERANCE	Use Cycle 32 to define the tolerance
4 CYCL DEF 32.1 T0.05	
5 CYCL DEF 32.2 HSC-MODE:1	
6 CYCL DEF 14.0 CONTOUR	Use Cycle 14 to refer to the contour in LBL1
7 CYCL DEF 14.1 CONTOUR LABEL1	
8 CYCL DEF 292 CONTOUR. TURNG. INTRP.	Define Cycle 292
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q546=+3 ;CHANGE TOOL DIRECTN.	
Q529=+0 ;MACHINING OPERATION	
Q221=+0 ;SURFACE OVERSIZE	
Q441=+1 ;INFEED	
Q449=+15000 ;FEED RATE	
Q491=+15 ;CONTOUR START RADIUS	
Q357=+2 ;CLEARANCE TO SIDE	
Q445=+50 ;CLEARANCE HEIGHT	
9 L Z+50 R0 FMAX M3	Pre-position in the tool axis, spindle on
10 L X+0 Y+0 R0 FMAX M99	Pre-position in the working plane to the center of rotation, call the cycle
11 LBL 1	LBL1 contains the contour
12 L Z+2 X+15	
13 L Z-5	
14 L Z-7 X+19	
15 RND R3	
16 L Z-15	
17 RND R2	
18 L X+27	

19 LBL 0	
20 M30	End of program
21 END PGM 2 MM	

12

Cycles: Turning

12.1 Turning Cycles (software option 50)

Overview

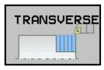
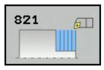
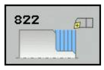
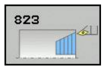
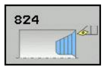
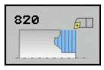
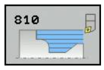
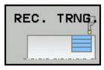
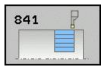
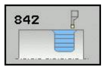
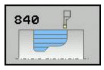
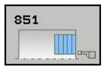
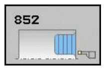
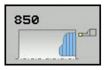
Defining turning cycles:

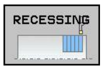
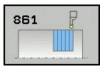
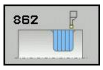
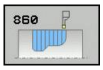
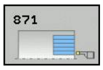
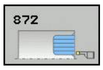
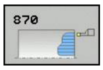
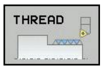
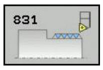
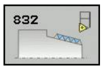
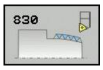
CYCL DEF
- The soft-key row shows the available groups of cycles

TURNING
- Select the menu for cycle group **TURNING**
- Select cycle group, e.g. cycles for longitudinal turning
- Select cycle, e.g. TURN SHOULDER, LONGITUDINAL

The TNC offers the following cycles for turning operations:

Cycle group	Cycle	Soft key	Page
Special cycles		SPECIAL CYCLES	
	ADAPT ROTARY COORDINATE SYSTEM(Cycle 800, DIN/ISO: G800)	800	322
	RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801)	801	328
	GEAR HOBBING (Cycle 880, DIN/ISO: G880)	880	425
	CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)	892	430
Cycles for longitudinal turning		LONGITDNL	329
	TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811)	811	330
	TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812)	812	333
	TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813)	813	337
	TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)	814	340
	TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)	810	344
	TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)	815	348

Cycle group	Cycle	Soft key	Page
Cycles for transverse turning			329
	TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)		352
	TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)		355
	TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)		359
	TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)		362
	TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)		366
	TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)		348
Cycles for recessing			
	SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)		370
	RADIAL RECESSING EXTENDED(Cycle 842, DIN/ISO: G842)		373
	RECESSING CONTOUR RADIAL(Cycle 840, DIN/ISO: G840)		378
	SIMPLE AXIAL RECESSING (Cycle 851, DIN/ISO: G851)		382
	AXIAL RECESSING EXTENDED(Cycle 852, DIN/ISO: G852)		385
	AXIAL RECESSING(Cycle 850, DIN/ISO: G850)		390

Cycle group	Cycle	Soft key	Page
Cycles for recessing			
	RADIAL RECESSING (Cycle 861, DIN/ISO: G861)		394
	RADIAL RECESSING EXTENDED(Cycle 862, DIN/ ISO: G862)		397
	RECESSING CONTOUR RADIAL(Cycle 860, DIN/ ISO: G860)		401
	AXIAL RECESSING (Cycle 871, DIN/ISO: G871)		405
	AXIAL RECESSING EXTENDED(Cycle 872, DIN/ ISO: G872)		407
	AXIAL RECESSING(Cycle 870, DIN/ISO: G870)		411
Cycles for thread turning			
	THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)		414
	THREAD EXTENDED (Cycle 832, DIN/ISO: G832)		417
	CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ ISO: G830)		421

Working with turning cycles

> You can only use turning cycles in Turning mode **FUNCTION MODE TURN**.

In turning cycles the TNC takes into account the cutting geometry (**TO, RS, P-ANGLE, T-ANGLE**) of the tool so that damage to the defined contour elements is prevented. The TNC outputs a warning if complete machining of the contour with the active tool is not possible.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the TNC detects the machining position (inside/outside machining) via the starting position or tool position when the cycle is called. In some cycles you can also enter the machining position directly in the cycle. After modifying the machining position, check the tool position and rotation direction.

If you program **M136** before a cycle, the TNC interprets feed rate values in the cycle in mm/rev., and without **M136** in mm/min.

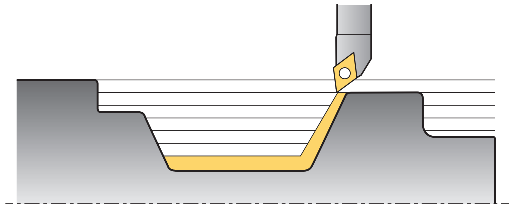
If turning cycles are executed during inclined machining (**M144**), the angles of the tool to the contour change. The TNC automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

Some cycles machine contours that you have written in a subprogram. You program these contours with plain-language path functions or FK functions. Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

You must call turning cycles 880 and 81x - 87x with **CYCL CALL** or **M99**. Before calling a cycle, be sure to program:

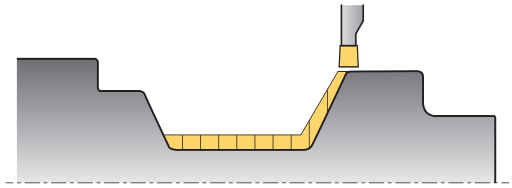
- Turning mode **FUNCTION MODE TURN**
- Tool call **TOOL CALL**
- Direction of rotation of turning spindle, e.g. **M303**
- Selection of speed/cutting speed **FUNCTION TURNDATA SPIN**
- If you use feed rate per revolution mm/rev., **M136**
- Tool positioning to suitable starting point e.g. **L X+130 Y+0 R0 FMAX**
- Adaptation of coordinate system and align tool **CYCL DEF 800 ADAPT ROTARY COORDINATE SYSTEM**

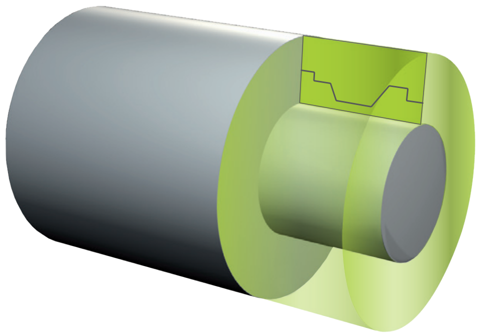
Blank form update (FUNCTION TURNDATA)



During turning operations workpieces must often be machined with several tools. Often a contour element cannot be completely finished because the tool form does not permit this (e.g. with a back cut). In this case, single sub-areas have to be reworked with other tools. The TNC detects the already machined areas with the blank form update and adapts all approach and departure paths to the specific, current machining situation. With the shorter machining paths, traverses in the air are avoided to significantly reduce machining time.

To activate the blank form update, program the **TURNDATA BLANK** function and link to a program or subprogram with a workpiece blank specification. The workpiece blank defined in **TURNDATA BLANK** determines the area to be machined with the blank form update. **TURNDATA BLANK OFF** deactivates blank form update.





> **!** The TNC optimizes machining areas and approach motions with blank form update. The TNC takes into account the specific tracked workpiece blank for approach and departure paths. If parts of the finished part extend beyond the workpiece blank, this may damage the workpiece and tool.

> Blank form update is only possible with cycle machining in turning mode (**FUNCTION MODE TURN**).
>
> You must define a closed contour as the workpiece blank for the blank form update (start position = end position). The workpiece blank corresponds to the cross-section of a rotationally symmetrical body.

The TNC has various options for defining the blank:

Workpiece blank definition	Soft key
Deactivate blank form update **TURNDATA BLANK OFF**: No input	BLANK OFF
Workpiece blank definition in a program: Enter name of the file	BLANK <FILE>
Workpiece blank definition in a program: Enter the string parameter with the program name	BLANK <FILE>=QS
Workpiece blank definition in a subprogram: Enter the number of the subprogram	BLANK LBL NR
Workpiece blank definition in a subprogram: Enter the name of the subprogram	BLANK LBL NAME
Workpiece blank definition in a subprogram: Enter the string parameter with the subprogram name	BLANK LBL QS

Activate blank form update and define workpiece blank:

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

TURNING PROGRAM FUNCTIONS
- Select the menu for **TURNING PROGRAM FUNCTIONS**

FUNCTION TURNDATA
- Select **BASIC FUNCTIONS**

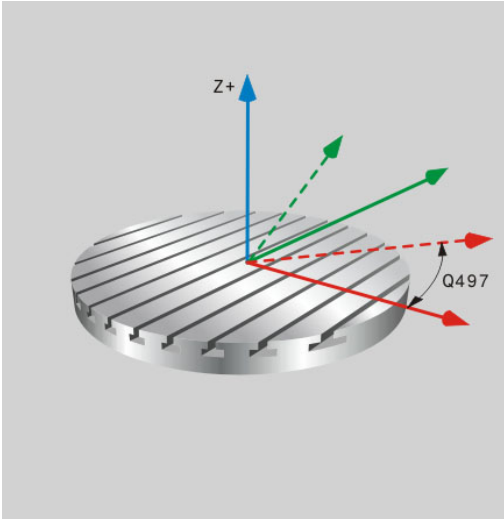
TURNDATA BLANK
- Select the function for blank form update

NC syntax

```
11 FUNCTION TURNDATABLANK LBL 20
```

12.2 ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800)

Application



This function must be adapted to the TNC by your machine manufacturer. Refer to your machine manual.

You need to position the tool appropriately with respect to the turning spindle, in order to be able to perform a turning operation. You can use Cycle **800 ADAPT ROTARY COORDINATE SYSTEM** for this.

The angle of incidence between the tool and the turning spindle is important for turning operations, for example, in order to machine contours with undercuts. Cycle 800 provides various possibilities for aligning the coordinate system for an inclined machining operation:

- If you have already positioned the tilting axis for an inclined machining operation, you can use Cycle 800 to align the coordinate system relative to the position of the tilting axes (**Q530=0**).
- Cycle 800 uses the angle of incidence Q531 to calculate the required tilting axis angle. Depending on the strategy selected in parameter **INCLINED MACHINING Q530**, the TNC positions the tilting axis with (**Q530=1**) or without compensating movement (**Q530=2**).
- Cycle 800 uses the angle of incidence **Q531** to calculate the required tilting axis angle, but does not perform any movements for positioning the tilting axis (**Q530=3**). You need to position the tilting axis to the calculated values Q120 (A axis), Q121 (B axis) and Q122 (C axis) after the cycle.

If you change the position of a tilting axis, you need to run Cycle 800 again to align the coordinate system.

If the axis of the milling spindle and the axis of the turning spindle are aligned parallel to each other, you can use **PRECESSION ANGLE Q497** to define any desired rotation of the coordinate system around the spindle axis (Z axis). This may be necessary if you have to bring the tool into a specific position due to space restrictions or if you want to improve your ability to observe a machining process. If the axis of the turning spindle is not aligned parallel to the axis of the milling spindle, only two precession angles are useful for machining. The TNC selects the angle that is closest to the input value **Q497**.

Cycle 800 positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can also use the mirrored tool (**REVERSE TOOL Q498**), thereby moving the position of the milling spindle by 180°. In this way you can use a tool for both inside and outside machining. Position the cutting edge at the center of the turning spindle using a positioning block, such as **L Y +0 R0 FMAX**.

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of rotation is aligned with the axis of the turning spindle (e.g. if large or rotationally non-symmetrical workpieces are being used). The **Q535** eccentric turning function in Cycle 800 enables you to perform turning operations in such cases as well.

During eccentric turning more than one linear axis is coupled to the turning spindle. The TNC compensates the eccentricity by performing circular compensating movements with the coupled linear axes.

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

High rotational speeds and large eccentricity require high feed rates of the linear axes to ensure that they move synchronously. If these feed rates are not maintained, the contour will be damaged. The TNC therefore generates an error message if 80 % of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the rotational speed.

Do not couple or decouple axes when the turning spindle is rotating. The TNC performs compensating movements during coupling and decoupling. Check for possible collisions.

> Turn a test cut before the actual machining operation to ensure that the required speeds can be attained.
>
> The linear axis positions resulting from the compensation are displayed by the TNC only in the ACTUAL value position display.

> 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.
>
> **Danger of collision!**
> Collision monitoring (DCM) is not active during eccentric turning. The TNC displays a corresponding warning during eccentric turning.

Effect

With Cycle 800 ADAPT ROTARY COORDINATE SYSTEM, the TNC aligns the workpiece coordinate system and orients the tool correspondingly. Cycle 800 is effective until it is reset by Cycle 801, or until Cycle 800 is defined again. Some cycle functions of Cycle 800 are additionally reset by other factors:

- The mirroring of the tool data (Q498 **REVERSE TOOL**) is reset by a **TOOL CALL**.
- The **ECCENTRIC TURNING** Q535 function is reset at the end of the program or due to cancelation of the program (internal stop).

Please note while programming:

The Cycle 800 ADAPT ROTARY COORDINATE SYSTEM is machine-dependent. Refer to your machine manual.

Software option 50 must be enabled

The tool must be clamped and measured in the correct position.

You can only mirror the tool data (Q498 **REVERSE TOOL**) when a turning tool is selected.

Check the orientation of the tool before machining.

Cycle 800 limits the maximum spindle speed during eccentric turning. Therefore, program Cycle 801 to reset Cycle 800, and reset the speed limitation with FUNCTION TURNDATA SPIN SMAX.

If you use the settings 1: MOVE, 2: TURN and 3: STAY in the parameter **Q530 INCLINED MACHINING**, the TNC activates the function **M144** (see also User's Manual for Inclined Turning).

Cycle parameters

- **PRECESSION ANGLE** Q497: Angle to which the TNC aligns the tool. Input range 0 to 359.9999
- **REVERSE TOOL** Q498: mirror tool for inside/ outside machining. Input range 0 and 1.
- **Inclined machining** Q530: Position the tilting axes for inclined machining:
  0: The position of the tilting axis (axis must have been positioned before) remains unchanged
  1: Position the tilting axis automatically, thereby orienting the tool tip (MOVE). The relative position between the tool and workpiece remains unchanged. The TNC performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
  3: Do not position the tilting axis. Position the tilting axes in a subsequent and separate positioning block (STAY). The TNC stores the position values in the parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis).
- **Angle of incidence** Q531: Angle of incidence for aligning the tool. Input range: -180° to +180°
- **Feed rate for positioning** Q532: Traverse speed of the tilting axis during automatic positioning. Input range 0.001 to 99999.999
- **Preferred direction** Q533: Selection of alternate possibilities of inclination. The angle of incidence you have defined is used by the TNC to calculate the appropriate positioning of the tilting axes present on the machine. In general there are always two possible solutions. Use the parameter Q533 to specify which solution the TNC should use:
  0: Select the solution using the shortest path
  -1: Select the solution in the negative direction
  +1: Select the solution in the positive direction

- **Eccentric turning** Q535: Couple the axes for the eccentric turning operation:
  0: Deactivate axis couplings
  1: Activate axis couplings The center of rotation is located at the active preset
  2: Activate axis couplings. The center of rotation is located at the active datum
  3: Do not change axis couplings.
- **Eccentric turning without stop** Q536: Interrupt program run before the axes are coupled:
  0: Stop before the axes are coupled again. In stopped condition, the TNC opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. Then press NC start to continue machining or press the **CANCEL** soft key to cancel machining
  1: Axes are coupled without stopping beforehand

12.3 RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801)

Please note while programming:

The Cycle 801 RESET ROTARY COORDINATE SYSTEM is machine-dependent. Refer to your machine manual.

With Cycle 801 RESET ROTARY COORDINATE SYSTEM you can reset the settings you have made with Cycle 800 ADAPT ROTARY COORDINATE SYSTEM.

Cycle 800 limits the maximum spindle speed during eccentric turning. Therefore, program Cycle 801 to reset Cycle 800, and reset the speed limitation with FUNCTION TURNDATA SPIN SMAX.

Effect

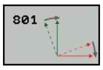
Cycle 801 resets the following settings you have programmed with Cycle 800:

- Precession angle Q497
- Reverse tool Q498

If you have executed the eccentric turning function with Cycle 800, the cycle limits the maximum spindle speed. To reset this, program FUNCTION TURNDATA SPIN SMAX in addition to Cycle 801.

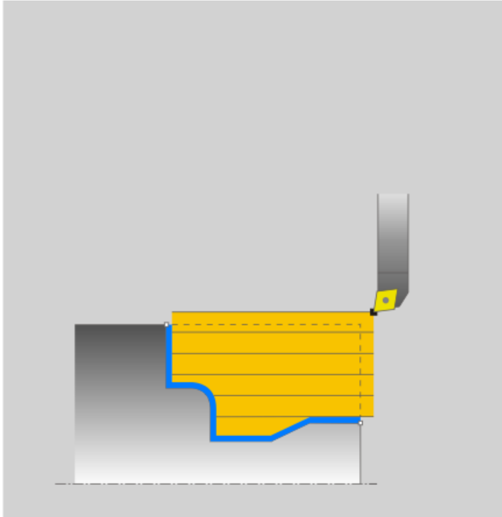
Cycle 801 does not orient the tool to the starting position. If a tool was oriented with Cycle 800, it remains in this position also after resetting.

Cycle parameters



- ▶ Cycle 801 does not have a cycle parameter. Finish the cycle input with the "END" key.

12.4 Fundamentals of Turning Cycles



The pre-positioning of the tool decisively affects the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when a cycle is called. When calculating the area to be machined, the TNC takes into account the starting point and the end point defined in the cycle or contour defined in the cycle. If the starting point lies in the area to be machined the TNC positions the tool beforehand in some cycles to set-up clearance.

The turning direction with 81x cycles is longitudinal to the rotary axis and lateral to the rotary axis with 82x cycles. The motions are contour-parallel in cycle 815.

The cycles can be used for inside and outside machining. The TNC takes the information for this from the position of the tool or the definition in the cycle (see "Working with turning cycles", page 319).

In cycles with freely defined contours (Cycles 810, 820 and 815), the programming direction of the contour determines the direction of machining.

In cycles for turning you can specify the machining strategies of roughing, finishing or complete machining.

> **!**
>
> **Caution: Danger to the workpiece and tool!**
>
> The turning cycles position the tool automatically to the starting point during finishing. The approach strategy is influenced by the position of the tool when the cycle is called. The decisive factor is whether the tool is located inside or outside an envelope contour when the cycle is called. The envelope contour is the programmed contour, enlarged by the set-up clearance.
>
> If the tool is within the envelope contour, the cycle positions the tool at the defined feed rate directly to the starting position. This can cause contour damage. Position the tool at a sufficient distance from the starting point to prevent the possibility of contour damage.
>
> If the tool is outside the envelope contour, positioning to the envelope contour is performed at rapid traverse, and at the programmed feed rate within the envelope contour.

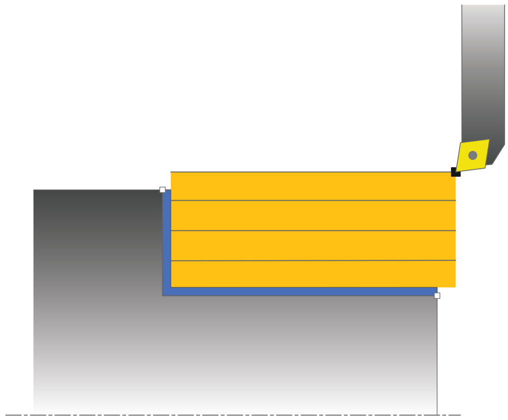
12.5 TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811)

Application

This cycle enables you to carry out longitudinal turning of right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle processes the area from the tool position to the end point defined in the cycle.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1. The TNC traverses the tool in the Z coordinate by the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. The TNC runs the paraxial infeed motion at rapid traverse.
3. The TNC finishes the finished part contour at the defined feed rate Q505.
4. The TNC returns the tool to set-up clearance at the defined feed rate.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

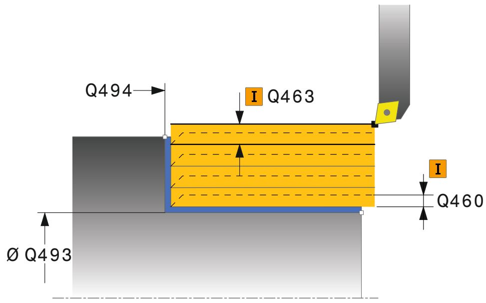
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

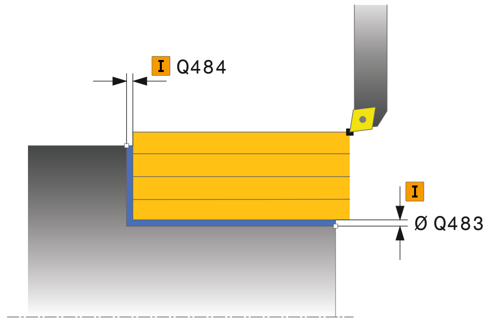
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°





NC blocks

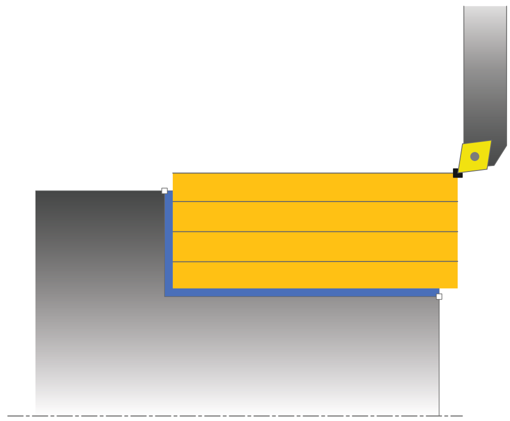
```
11 CYCL DEF 811 TURN SHOULDER LONG.
    Q215=+0    ;MACHINING OPERATION
    Q460=+2    ;SAFETY CLEARANCE
    Q493=+50   ;DIAMETER AT END OF CONTOUR
    Q494=-55   ;CONTOUR END IN Z
    Q463=+3    ;MAX. CUTTING DEPTH
    Q478=+0.3  ;ROUGHING FEED RATE
    Q483=+0.4  ;OVERSIZE FOR DIAMETER
    Q484=+0.2  ;OVERSIZE IN Z
    Q505=+0.2  ;FINISHING FEED RATE
    Q506=+0    ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.6 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812)

Application

This cycle enables you to run longitudinal turning of shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge



You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the starting point is within the area to be machined, the TNC positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

If the starting point lies in the area to be machined, the TNC positions the tool beforehand to set-up clearance in the Z coordinate.

1 The TNC runs the paraxial infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

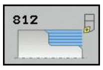
Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

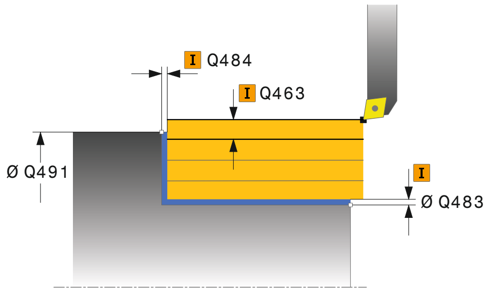
The tool position at cycle call (cycle starting point) affects the area to be machined.

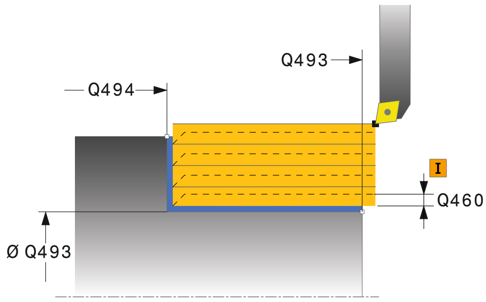
Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the contour starting point
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of circumferential surface** Q495: Angle between the circumferential surface and the rotary axis





- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of face** Q496: Angle between the face and the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

```
11 CYCL DEF 812 TURN SHOULDER
   LONG. EXTENDED.
   Q215=+0     ;MACHINING
               OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q491=+75    ;DIAMETER AT
               CONTOUR START
   Q492=+0     ;CONTOUR START IN Z
   Q493=+50    ;DIAMETER AT END OF
               CONTOUR
   Q494=-55    ;CONTOUR END IN Z
   Q495=+5     ;ANGLE OF CIRCUM.
               SURFACE
   Q501=+1     ;TYPE OF STARTING
               ELEMENT
   Q502=+0.5   ;SIZE OF STARTING
               ELEMENT
   Q500=+1.5   ;RADIUS OF CONTOUR
               EDGE
   Q496=+0     ;ANGLE OF FACE
   Q503=+1     ;TYPE OF END
               ELEMENT
   Q504=+0.5   ;SIZE OF END ELEMENT
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR
               DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q506=+0     ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

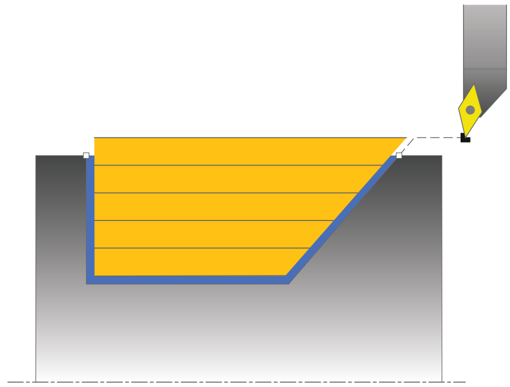
12.7 TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813)

Application

This cycle enables you to run longitudinal turning of shoulders with plunge elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q492 CONTOUR START IN Z**, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting the TNC runs the infeed with feed rate **Q478**. The return movements are then each at set-up clearance.

1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2 The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The TNC returns the tool at the defined feed rate by one infeed value.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

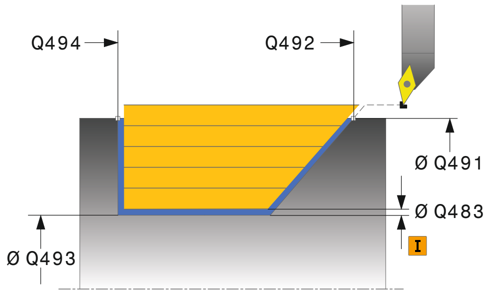
The tool position at cycle call (cycle starting point) affects the area to be machined.

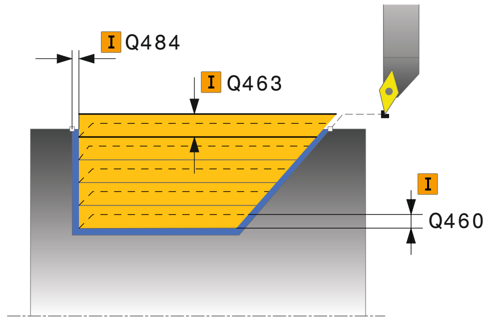
The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC.

Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the perpendicular to the rotary axis.
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°



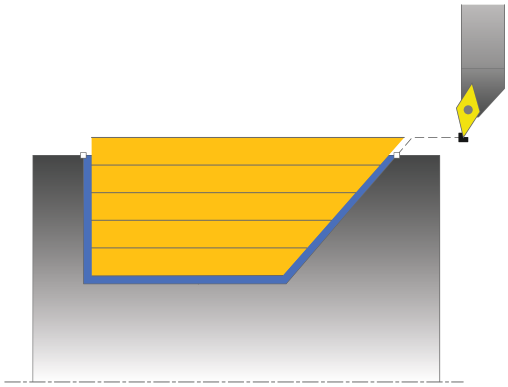


NC blocks

```
11 CYCL DEF 813 TURN,
   LONGITUDINAL PLUNGE
   Q215=+0    ;MACHINING
               OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q491=+75   ;DIAMETER AT
               CONTOUR START
   Q492=-10   ;CONTOUR START IN Z
   Q493=+50   ;DIAMETER AT END OF
               CONTOUR
   Q494=-55   ;CONTOUR END IN Z
   Q495=+70   ;ANGLE OF SIDE
   Q463=+3    ;MAX. CUTTING DEPTH
   Q478=+0.3  ;ROUGHING FEED RATE
   Q483=+0.4  ;OVERSIZE FOR
               DIAMETER
   Q484=+0.2  ;OVERSIZE IN Z
   Q505=+0.2  ;FINISHING FEED RATE
   Q506=+0    ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)

Application



This cycle enables you to run longitudinal turning of shoulders with plunge elements (undercuts). Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q492 CONTOUR START IN Z**, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting the TNC runs the infeed with feed rate **Q478**. The return movements are then each at set-up clearance.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1 The TNC runs the infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

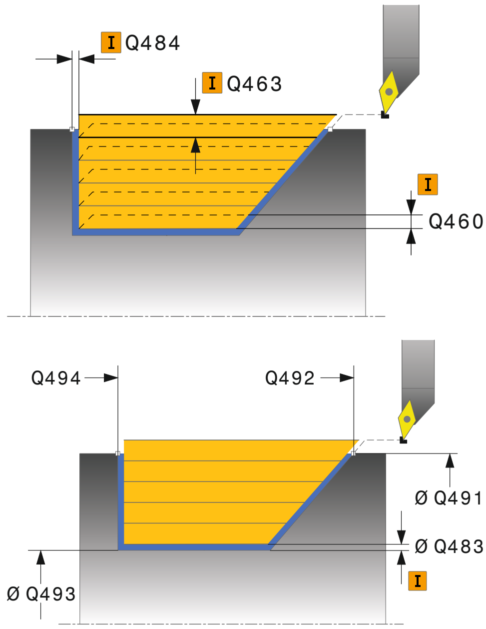
The tool position at cycle call (cycle starting point) affects the area to be machined.

The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC.

Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the perpendicular to the rotary axis.



- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of face** Q496: Angle between the face and the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

```
11 CYCL DEF 814 TURN, LONGITUDINAL PLUNGE EXT.
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q491=+75    ;DIAMETER AT CONTOUR START
   Q492=-10    ;CONTOUR START IN Z
   Q493=+50    ;DIAMETER AT END OF CONTOUR
   Q494=-55    ;CONTOUR END IN Z
   Q495=+70    ;ANGLE OF SIDE
   Q501=+1     ;TYPE OF STARTING ELEMENT
   Q502=+0.5   ;SIZE OF STARTING ELEMENT
   Q500=+1.5   ;RADIUS OF CONTOUR EDGE
   Q496=+0     ;ANGLE OF FACE
   Q503=+1     ;TYPE OF END ELEMENT
   Q504=+0.5   ;SIZE OF END ELEMENT
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q506=+0     ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

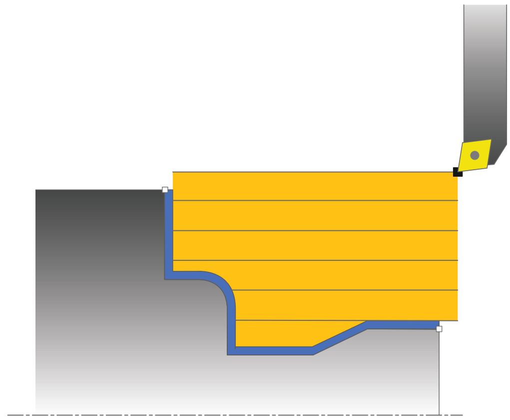
12.9 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)

Application

This cycle enables you to run longitudinal turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2 The TNC machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially with the defined feed rate **Q478**.
3 The TNC returns the tool at the defined feed rate by one infeed value.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle starting point) affects the area to be machined.

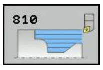
The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC.

Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

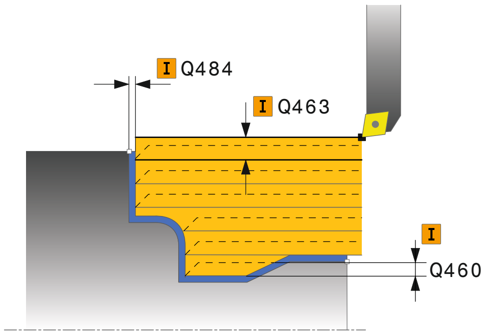
Also refer to the fundamentals of turning cycles (see page 329).

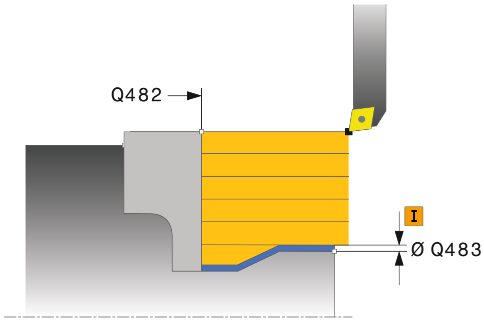
When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Reverse contour** Q499: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour





- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Plunging** Q487: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 810 TURN CONTOUR LONG.
    Q215=+0    ;MACHINING OPERATION
    Q460=+2    ;SAFETY CLEARANCE
    Q499=+0    ;REVERSE CONTOUR
    Q463=+3    ;MAX. CUTTING DEPTH
    Q478=+0.3  ;ROUGHING FEED RATE
    Q483=+0.4  ;OVERSIZE FOR DIAMETER
    Q484=+0.2  ;OVERSIZE IN Z
    Q505=+0.2  ;FINISHING FEED RATE
    Q487=+1    ;PLUNGE
    Q488=+0    ;PLUNGING FEED RATE
    Q479=+0    ;CUTTING LIMIT
    Q480=+0    ;LIMIT VALUE FOR DIAMETER
    Q482=+0    ;LIMIT VALUE IN Z
    Q506=+0    ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-35
20 RND R5
21 L X+50 Z-40
22 L Z-55
23 CC X+60 Z-55
24 C X+60 Z-60
25 L X+100
26 LBL 0
```

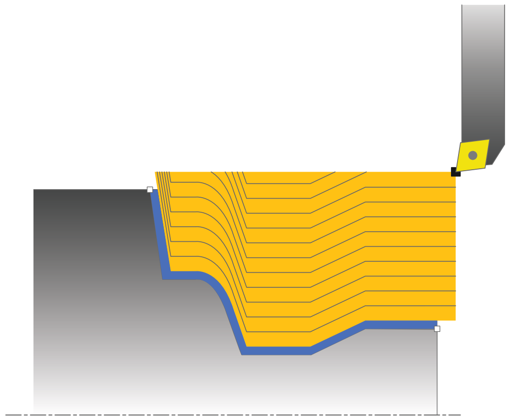
12.10 TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)

Application

This cycle enables you to machine workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning with roughing is contour-parallel.

The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2 The TNC machines the area between the starting position and end point. The cut is run contour-parallel with the defined feed rate **Q478**.
3 The TNC returns the tool at the defined feed rate back to the starting position in the X coordinate.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle starting point) affects the area to be machined.

The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC.

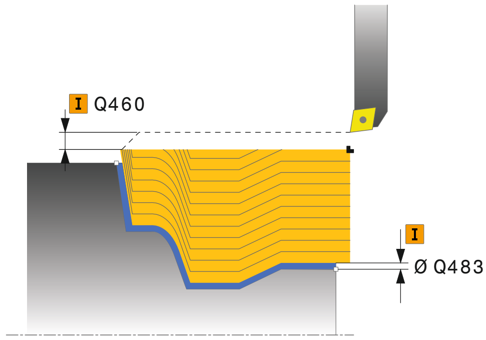
Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

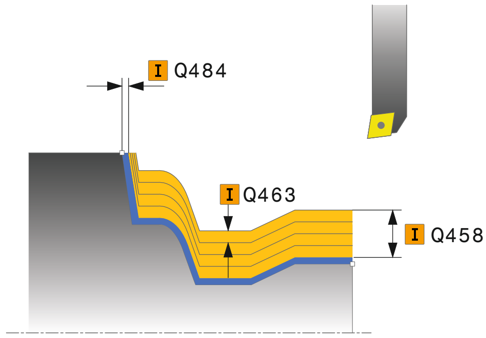
Also refer to the fundamentals of turning cycles (see page 329).

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Oversize for blank** Q485 (incremental): Contour-parallel oversize for the defined contour
- **Cutting lines** Q486: Define the type of cutting lines:
  0: Cuts with constant chip cross section
  1: Equidistant proportioning of cuts
- **Reverse contour** Q499: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999





- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

NC blocks

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 815 TURN CONTOUR-
   PARALLEL
   Q215=+0    ;MACHINING
              OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q485=+5    ;OVERSIZE ON BLANK
   Q486=+0    ;CUTTING LINES
   Q499=+0    ;REVERSE CONTOUR
   Q463=+3    ;MAX. CUTTING DEPTH
   Q483=+0.4  ;OVERSIZE FOR
              DIAMETER
   Q484=+0.2  ;OVERSIZE IN Z
   Q505=+0.2  ;FINISHING FEED RATE
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-35
20 RND R5
21 L X+50 Z-40
22 L Z-55
23 CC X+60 Z-55
24 C X+60 Z-60
25 L X+100
26 LBL 0
```

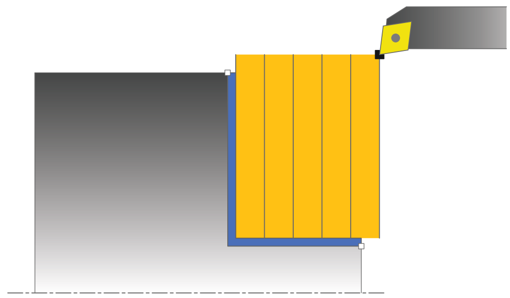
12.11 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)

Application

This cycle enables you to face turn right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle processes the area from the cycle starting point to the end point defined in the cycle.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate **Q478**.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1. The TNC traverses the tool in the Z coordinate by the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. The TNC runs the paraxial infeed motion at rapid traverse.
3. The TNC finishes the finished part contour at the defined feed rate **Q505**.
4. The TNC returns the tool to set-up clearance at the defined feed rate.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

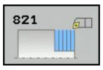
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

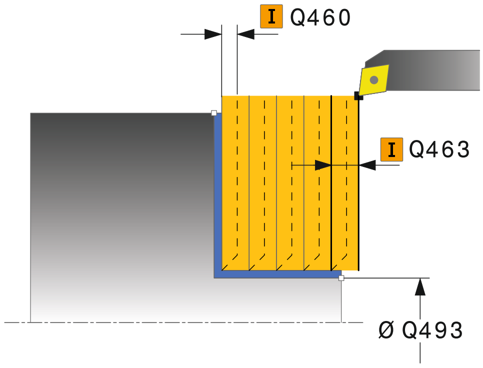
The tool position at cycle call (cycle starting point) affects the area to be machined.

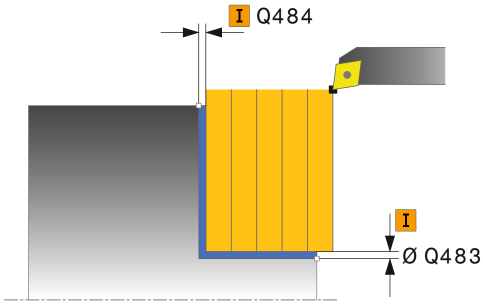
Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°





NC blocks

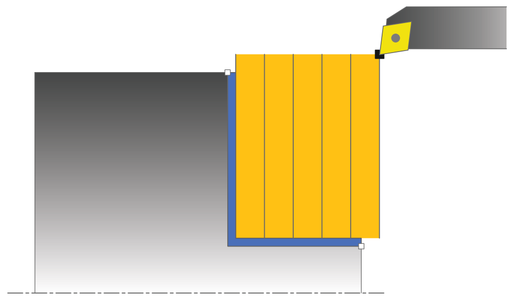
11 CYCL DEF 821 TURN SHOULDER FACE	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+30	;DIAMETER AT END OF CONTOUR
Q494=-5	;CONTOUR END IN Z
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

12.12 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)

Application

This cycle enables you to face turn shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge



You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the starting point is within the area to be machined, the TNC positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate **Q478**.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1. The TNC runs the paraxial infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

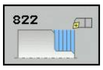
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

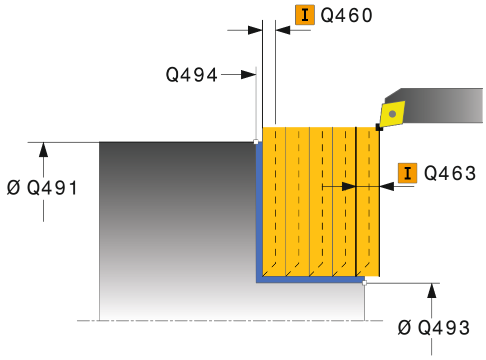
The tool position at cycle call (cycle starting point) affects the area to be machined.

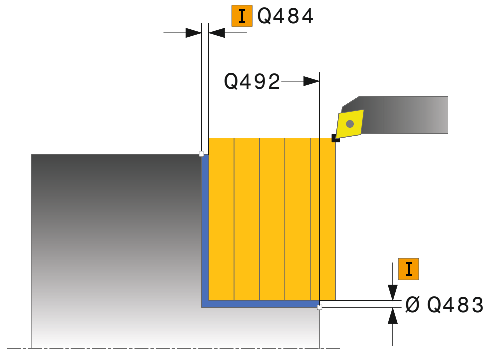
Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the contour starting point
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of face** Q495: Angle between the face and the rotary axis
- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius





- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of circumferential surface** Q496: Angle between the circumferential surface and the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

```
11 CYCL DEF 822 TURN SHOULDER
   FACE EXTENDED
   Q215=+0    ;MACHINING
              OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q491=+75   ;DIAMETER AT
              CONTOUR START
   Q492=+0    ;CONTOUR START IN Z
   Q493=+30   ;DIAMETER AT END OF
              CONTOUR
   Q494=-15   ;CONTOUR END IN Z
   Q495=+0    ;ANGLE OF FACE
   Q501=+1    ;TYPE OF STARTING
              ELEMENT
   Q502=+0.5 ;SIZE OF STARTING
              ELEMENT
   Q500=+1.5 ;RADIUS OF CONTOUR
              EDGE
   Q496=+5    ;ANGLE OF CIRCUM.
              SURFACE
   Q503=+1    ;TYPE OF END
              ELEMENT
   Q504=+0.5 ;SIZE OF END ELEMENT
   Q463=+3    ;MAX. CUTTING DEPTH
   Q478=+0.3 ;ROUGHING FEED RATE
   Q483=+0.4 ;OVERSIZE FOR
              DIAMETER
   Q484=+0.2 ;OVERSIZE IN Z
   Q505=+0.2 ;FINISHING FEED RATE
   Q506=+0    ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

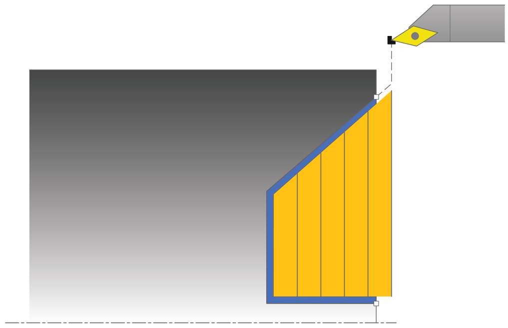
12.13 TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)

Application

This cycle enables you to face turn plunge elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.



Roughing cycle run

In undercutting the TNC runs the infeed with feed rate **Q478**. The return movements are then each at set-up clearance.

1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2 The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The TNC returns the tool at the defined feed rate **Q478** by one infeed value.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The TNC runs the infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

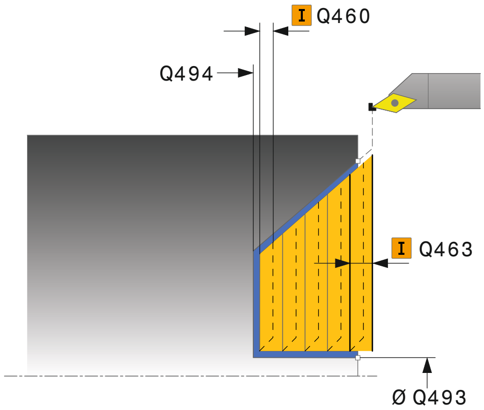
The tool position at cycle call (cycle starting point) affects the area to be machined.

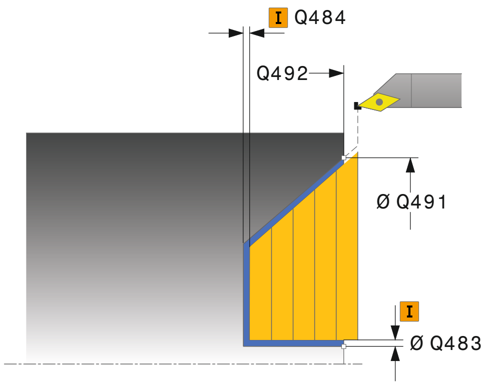
The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements.
If complete machining with the active tool is not possible, a warning is output by the TNC.

Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°



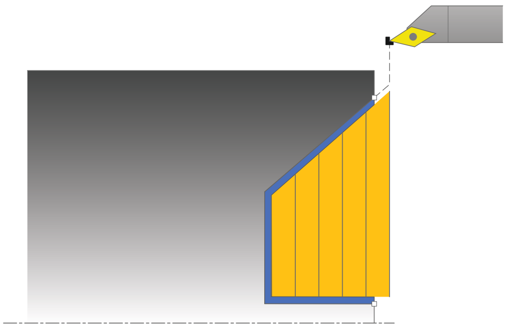


NC blocks

```
11 CYCL DEF 823 TURN, TRANSVERSE PLUNGE
    Q215=+0    ;MACHINING OPERATION
    Q460=+2    ;SAFETY CLEARANCE
    Q491=+75   ;DIAMETER AT CONTOUR START
    Q492=+0    ;CONTOUR START IN Z
    Q493=+20   ;DIAMETER AT END OF CONTOUR
    Q494=-5    ;CONTOUR END IN Z
    Q495=+60   ;ANGLE OF SIDE
    Q463=+3    ;MAX. CUTTING DEPTH
    Q478=+0.3  ;ROUGHING FEED RATE
    Q483=+0.4  ;OVERSIZE FOR DIAMETER
    Q484=+0.2  ;OVERSIZE IN Z
    Q505=+0.2  ;FINISHING FEED RATE
    Q506=+0    ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.14 TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)

Application



This cycle enables you to face turn plunge elements (undercuts). Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

In undercutting the TNC runs the infeed with feed rate **Q478**. The return movements are then each at set-up clearance.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate.
3. The TNC returns the tool at the defined feed rate **Q478** by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The TNC runs the infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

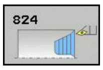
Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle starting point) affects the area to be machined.

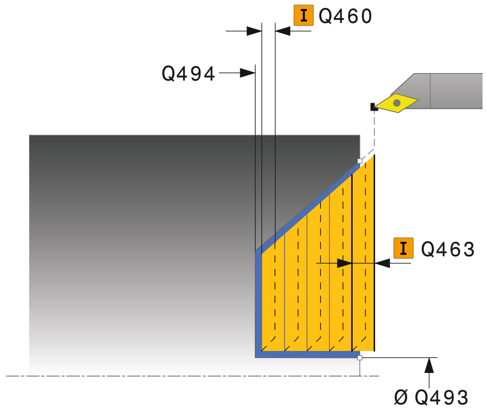
The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC.

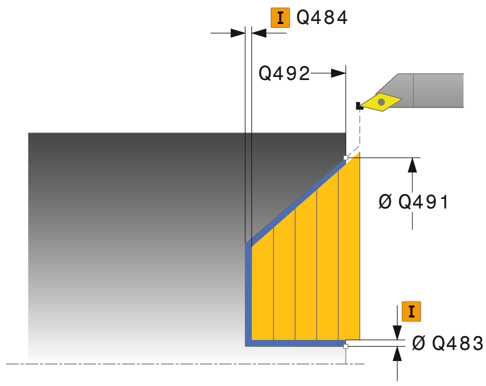
Also refer to the fundamentals of turning cycles (see page 329).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Diameter at contour start** Q491: X coordinate of the starting point for the plunging path (diameter value)
- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius





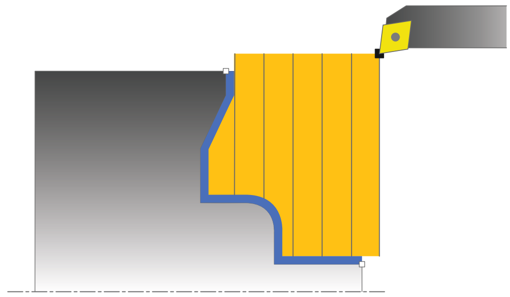
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

```
11 CYCL DEF 824 TURN, TRANSVERSE
   PLUNGE EXT.
   Q215=+0    ;MACHINING
              OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q491=+75   ;DIAMETER AT
              CONTOUR START
   Q492=+0    ;CONTOUR START IN Z
   Q493=+20   ;DIAMETER AT END OF
              CONTOUR
   Q494=-10   ;CONTOUR END IN Z
   Q495=+70   ;ANGLE OF SIDE
   Q501=+1    ;TYPE OF STARTING
              ELEMENT
   Q502=+0.5  ;SIZE OF STARTING
              ELEMENT
   Q500=+1.5  ;RADIUS OF CONTOUR
              EDGE
   Q496=+0    ;ANGLE OF FACE
   Q503=+1    ;TYPE OF END
              ELEMENT
   Q504=+0.5  ;SIZE OF END ELEMENT
   Q463=+3    ;MAX. CUTTING DEPTH
   Q478=+0.3  ;ROUGHING FEED RATE
   Q483=+0.4  ;OVERSIZE FOR
              DIAMETER
   Q484=+0.2  ;OVERSIZE IN Z
   Q505=+0.2  ;FINISHING FEED RATE
   Q506=+0    ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.15 TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)

Application



This cycle enables you to face turn workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2. The TNC machines the area between the starting position and the end point in traverse direction. The transverse cut is run paraxially with the defined feed rate **Q478**.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The TNC runs the infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle starting point) affects the area to be machined.

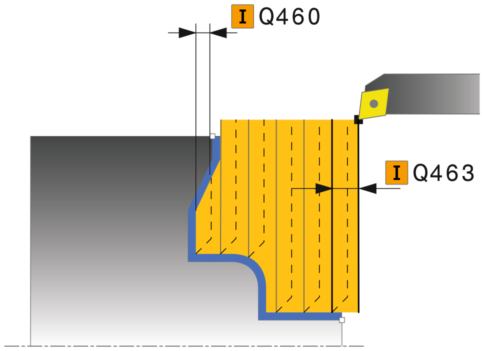
The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC.

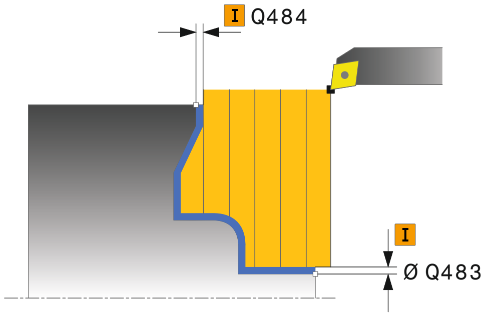
Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

Also refer to the fundamentals of turning cycles (see page 329).

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Cycle parameters





- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.
- **Reverse contour** Q499: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Plunging** Q487: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 820 TURN CONTOUR FACE
Q215=+0 ;MACHINING OPERATION
Q460=+2 ;SAFETY CLEARANCE
Q499=+0 ;REVERSE CONTOUR
Q463=+3 ;MAX. CUTTING DEPTH
Q478=+0.3 ;ROUGHING FEED RATE
Q483=+0.4 ;OVERSIZE FOR DIAMETER
Q484=+0.2 ;OVERSIZE IN Z
Q505=+0.2 ;FINISHING FEED RATE
Q487=+1 ;PLUNGE
Q488=+0 ;PLUNGING FEED RATE
Q479=+0 ;CUTTING LIMIT
Q480=+0 ;LIMIT VALUE FOR DIAMETER
Q482=+0 ;LIMIT VALUE IN Z
Q506=+0 ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+75 Z-20
17 L X+50
18 RND R2
19 L X+20 Z-25
20 RND R2
21 L Z+0
22 LBL 0

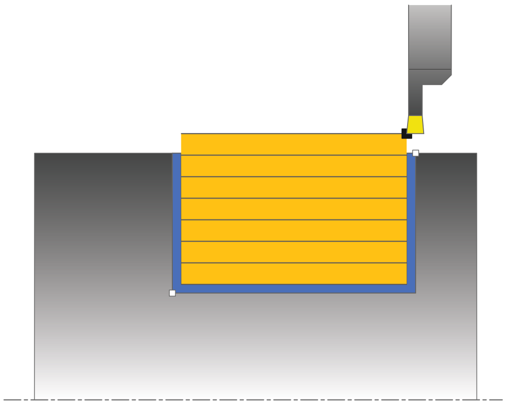
12.16 SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)

Application

This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. The cycle processes only the area from the cycle starting point to the end point defined in the cycle.

1 From the cycle starting point, the TNC recesses until the first plunging depth.
2 The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction **Q507=1** was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The TNC repeats this process (2 to 4) until the slot depth is reached.
7 The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes the slot floor at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC positions the tool back at rapid traverse to the cycle starting point.

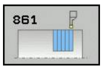
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

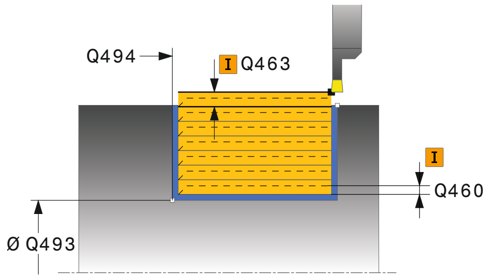
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

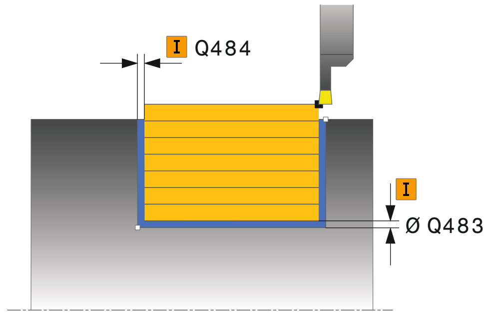
From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.
- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.



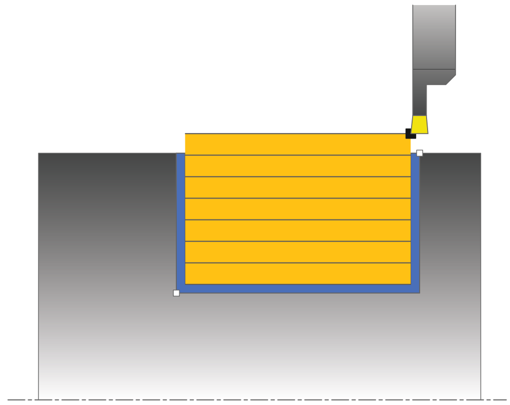


NC blocks

11 CYCL DEF 841 RECESS TURNG. SIMPLE R.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT END OF CONTOUR
Q494=-50	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+2	;MAX. CUTTING DEPTH
Q507=+0	;MACHINING DIRECTION
Q508=+0	;OFFSET WIDTH
Q509=+0	;DEPTH COMPENSATION
Q488=+0	;PLUNGING FEED RATE
12 L X+75 Y+0 Z-25 FMAX M303	
13 CYCL CALL	

12.17 RADIAL RECESSING EXTENDED (Cycle 842, DIN/ISO: G842)

Application



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q491 DIAMETER AT CONTOUR START**, the TNC positions the tool in the X coordinate to **Q491** and begins the cycle there.

1. From the cycle starting point, the TNC recesses until the first plunging depth.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The TNC repeats this process (2 to 4) until the slot depth is reached.
7. The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q491 DIAMETER AT CONTOUR START**, the TNC positions the tool in the X coordinate to **Q491** and begins the cycle there.

1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate Q505.
3 The TNC finishes the slot floor at the defined feed rate. If a radius for contour edges Q500 was specified, the TNC finishes the complete slot in one pass.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot side.
6 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7 The TNC positions the tool back at rapid traverse to the cycle starting point.

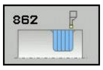
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

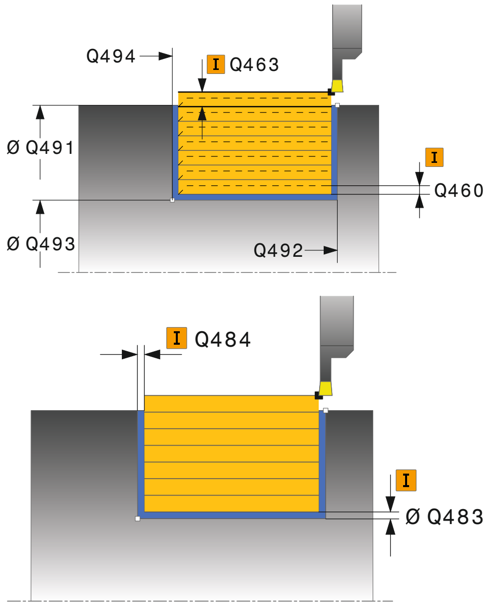
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the contour starting point
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle between the side at the contour starting point and the perpendicular to the rotary axis



- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of second side** Q496: Angle between the side at the contour end point and the perpendicular to the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

NC blocks

```
11 CYCL DEF 842 RADIAL RECESSING EXTENDED
   Q215=+0    ;MACHINING OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q491=+75   ;DIAMETER AT CONTOUR START
   Q492=-20   ;CONTOUR START IN Z
   Q493=+50   ;DIAMETER AT END OF CONTOUR
   Q494=-50   ;CONTOUR END IN Z
   Q495=+5    ;ANGLE OF SIDE
   Q501=+1    ;TYPE OF STARTING ELEMENT
   Q502=+0.5  ;SIZE OF STARTING ELEMENT
   Q500=+1.5  ;RADIUS OF CONTOUR EDGE
   Q496=+5    ;ANGLE OF SECOND SIDE
   Q503=+1    ;TYPE OF END ELEMENT
   Q504=+0.5  ;SIZE OF END ELEMENT
   Q478=+0.3  ;ROUGHING FEED RATE
   Q483=+0.4  ;OVERSIZE FOR DIAMETER
   Q484=+0.2  ;OVERSIZE IN Z
   Q505=+0.2  ;FINISHING FEED RATE
   Q463=+2    ;MAX. CUTTING DEPTH
   Q507=+0    ;MACHINING DIRECTION
   Q508=+0    ;OFFSET WIDTH
   Q509=+0    ;DEPTH COMPENSATION
   Q488=+0    ;PLUNGING FEED RATE
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.

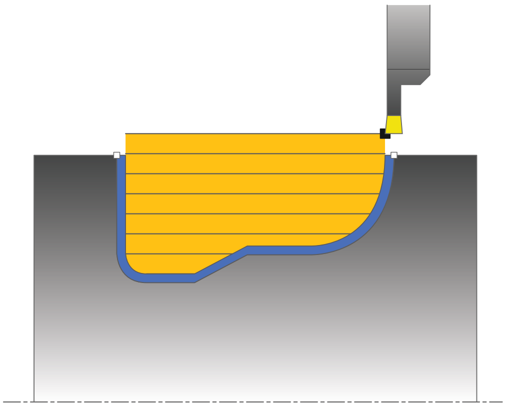
12.18 RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)

Application

This cycle enables you to recess right-angled slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the X coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the X coordinate to the contour starting point and begins the cycle there.

1 The TNC positions the tool at rapid traverse in the Z coordinate (first cut-in position).
2 The TNC recesses until the first plunging depth.
3 The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
4 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5 If only one machining direction **Q507=1** was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides. .
6 The tool recesses to the next plunging depth.
7 The TNC repeats this process (2 to 4) until the slot depth is reached.
8 The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

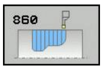
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side walls of the slot at the defined feed rate **Q505**.
3. The TNC finishes the slot floor at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

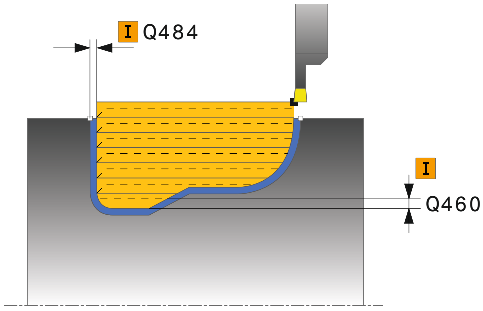
> **!**
>
> The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.
>
> The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

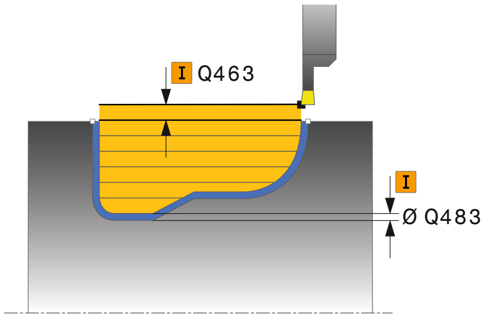
> ⇨
>
> Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
>
> The tool position at cycle call defines the size of the area to be machined (cycle starting point).
>
> Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.
>
> When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
>
> From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction





- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.
- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Reverse contour** Q499: Machining direction:
  0: Machining in contour direction
  1: Machining opposite the contour direction

NC blocks

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 840 RECESS TURNG. RAD
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q478=+0.3   ;ROUGHING FEED RATE
    Q488=+0     ;PLUNGING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q479=+0     ;CUTTING LIMIT
    Q480=+0     ;LIMIT VALUE FOR DIAMETER
    Q482=+0     ;LIMIT VALUE IN Z
    Q463=+2     ;MAX. CUTTING DEPTH
    Q507=+0     ;MACHINING DIRECTION
    Q508=+0     ;OFFSET WIDTH
    Q509=+0     ;DEPTH COMPENSATION
    Q499=+0     ;REVERSE CONTOUR
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z-10
17 L X+40 Z-15
18 RND R3
19 CR X+40 Z-35 R+30 DR+
18 RND R3
20 L X+60 Z-40
21 LBL 0
```

12.19 SIMPLE AXIAL RECESSING (Cycle 851, DIN/ISO: G851)

Application

This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. The cycle processes the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the TNC recesses until the first plunging depth.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The TNC repeats this process (2 to 4) until the slot depth is reached.
7. The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3 The TNC finishes the slot floor at the defined feed rate.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot side.
6 The TNC finishes the side wall of the slot at the defined feed rate Q505.
7 The TNC positions the tool back at rapid traverse to the cycle starting point.

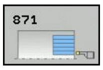
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

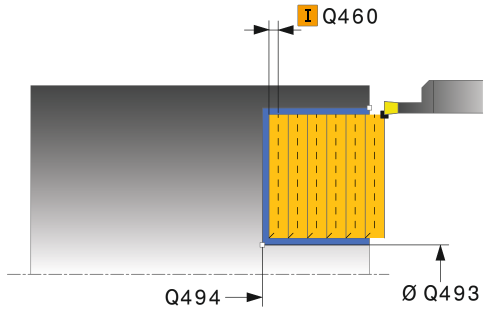
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

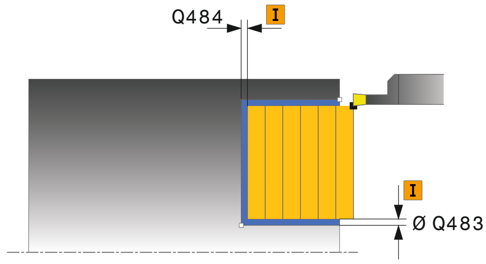
From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.
- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.



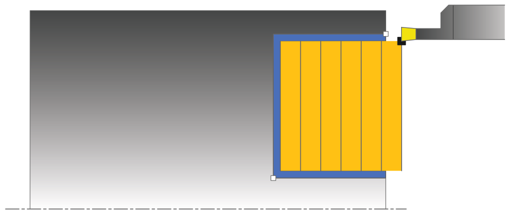


NC blocks

```
11 CYCL DEF 851 RECESS TURNG,
   SIMPLE AXIAL
   Q215=+0     ;MACHINING
                OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q493=+50    ;DIAMETER AT END OF
                CONTOUR
   Q494=-10    ;CONTOUR END IN Z
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR
                DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q463=+2     ;MAX. CUTTING DEPTH
   Q507=+0     ;MACHINING DIRECTION
   Q508=+0     ;OFFSET WIDTH
   Q509=+0     ;DEPTH COMPENSATION
   Q488=+0     ;PLUNGING FEED RATE
12 L X+65 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.20 AXIAL RECESSING EXTENDED (Cycle 852, DIN/ISO: G852)

Application



This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q492 CONTOUR START IN Z**, the TNC positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. From the cycle starting point, the TNC recesses until the first plunging depth.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The TNC repeats this process (2 to 4) until the slot depth is reached.
7. The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q492 CONTOUR START IN Z**, the TNC positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3 The TNC finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the TNC finishes the complete slot in one pass.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot side.
6 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7 The TNC positions the tool back at rapid traverse to the cycle starting point.

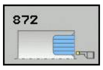
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

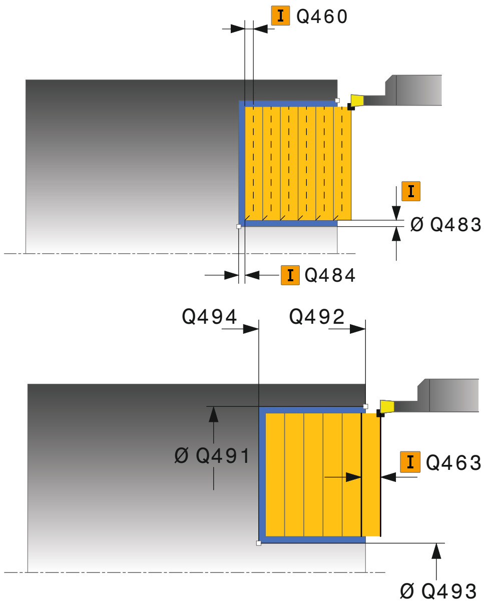
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the contour starting point
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle between the side at the contour starting point and the parallel line to the rotary axis



- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of second side** Q496: Angle between the side at the contour end point and the parallel line to the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

NC blocks

11 CYCL DEF 852 RECESS TURNG. AXIAL EXTENDED	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT END OF CONTOUR
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF SECOND SIDE
Q503=+1	;TYPE OF END ELEMENT
Q504=+0.5	;SIZE OF END ELEMENT
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+2	;MAX. CUTTING DEPTH
Q507=+0	;MACHINING DIRECTION
Q508=+0	;OFFSET WIDTH
Q509=+0	;DEPTH COMPENSATION
Q488=+0	;PLUNGING FEED RATE
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.

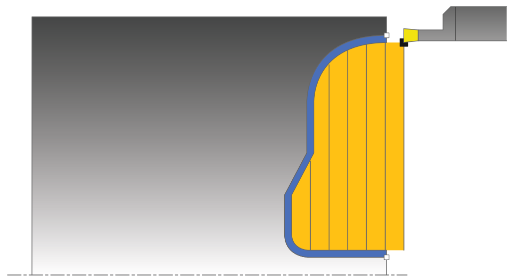
12.21 AXIAL RECESSING (Cycle 850, DIN/ISO: G850)

Application

This cycle enables you to recess right-angled slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1. The TNC positions the tool at rapid traverse in the X coordinate (first cut-in position).
2. The TNC recesses until the first plunging depth.
3. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate **Q478**.
4. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5. If only one machining direction **Q507=1** was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides. .
6. The tool recesses to the next plunging depth.
7. The TNC repeats this process (2 to 4) until the slot depth is reached.
8. The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
9. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.

1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side walls of the slot at the defined feed rate **Q505**.
3 The TNC finishes the slot floor at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

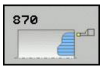
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

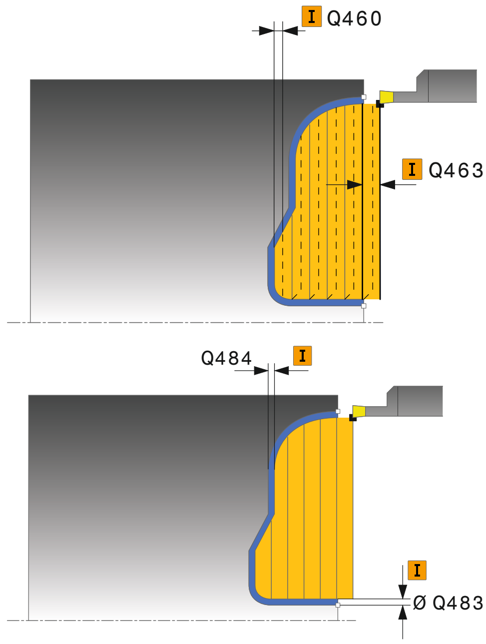
When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction



- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.
- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Reverse contour** Q499: Machining direction:
  0: Machining in contour direction
  1: Machining opposite the contour direction

NC blocks

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 850 RECESS TURNG.
   AXIAL
   Q215=+0    ;MACHINING
               OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q478=+0.3 ;ROUGHING FEED RATE
   Q483=+0.4 ;OVERSIZE FOR
               DIAMETER
   Q484=+0.2 ;OVERSIZE IN Z
   Q505=+0.2 ;FINISHING FEED RATE
   Q479=+0    ;CUTTING LIMIT
   Q480=+0    ;LIMIT VALUE FOR
               DIAMETER
   Q482=+0    ;LIMIT VALUE IN Z
   Q463=+2    ;MAX. CUTTING DEPTH
   Q507=+0    ;MACHINING DIRECTION
   Q508=+0    ;OFFSET WIDTH
   Q509=+0    ;DEPTH COMPENSATION
   Q499=+0    ;REVERSE CONTOUR
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-15
20 L Z+0
21 LBL 0
```

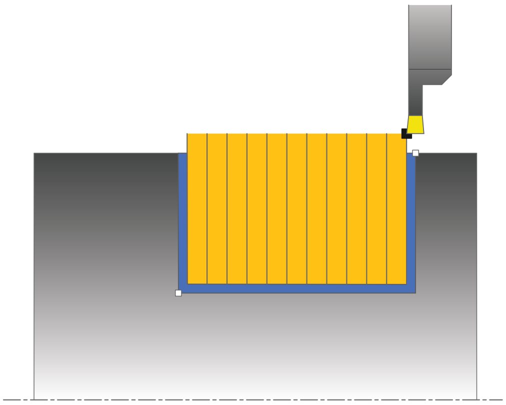
12.22 RADIAL RECESSING (Cycle 861, DIN/ISO: G861)

Application

This cycle enables you to radially cut in right-angled slots.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle processes only the area from the cycle starting point to the end point defined in the cycle.

1. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2. The TNC cuts the area between the starting position and the end point in axial direction at the defined feed rate **Q478**.
3. The TNC positions the tool back at rapid traverse to the beginning of cut.
4. The TNC repeats this process (1 to 3) until the slot width is reached.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

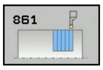
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes half the slot width at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC finishes half the slot width at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

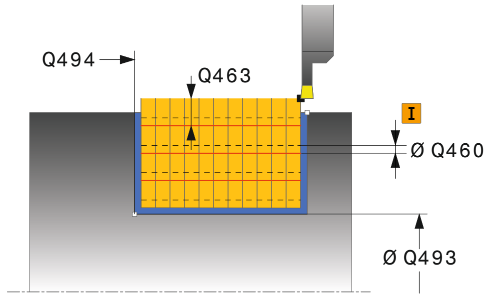
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

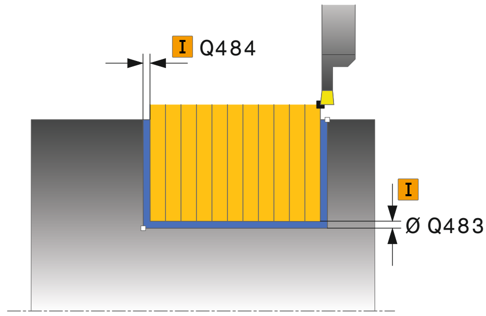
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Limit to depth** Q463: Max. recess depth per cut



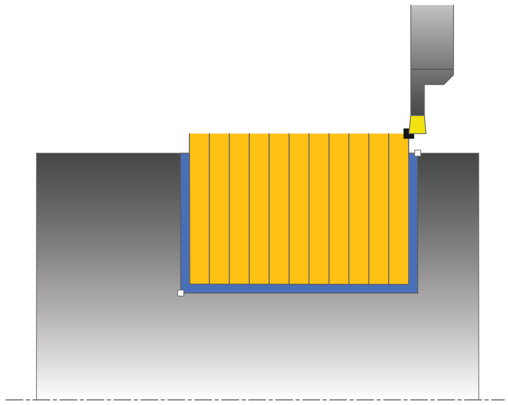


NC blocks

```
11 CYCL DEF 861 RADIAL RECESSING
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q493=+50    ;DIAMETER AT END OF CONTOUR
    Q494=-50    ;CONTOUR END IN Z
    Q478=+0.3   ;ROUGHING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q463=+0     ;LIMIT TO DEPTH
12 L X+75 Y+0 Z-25 FMAX M303
13 CYCL CALL
```

12.23 RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)

Application



This cycle enables you to radially cut in slots. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

1. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2. The TNC cuts the area between the starting position and the end point in axial direction at the defined feed rate **Q478**.
3. The TNC positions the tool back at rapid traverse to the beginning of cut.
4. The TNC repeats this process (1 to 3) until the slot width is reached.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

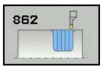
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes half the slot width at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC finishes half the slot width at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

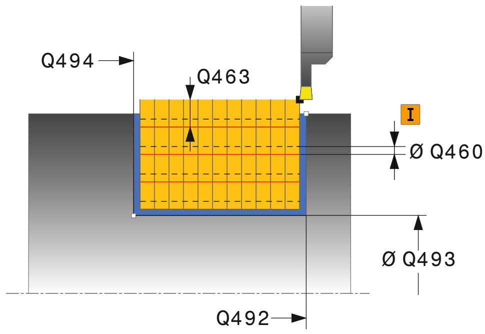
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The tool position at cycle call defines the size of the area to be machined (cycle starting point).

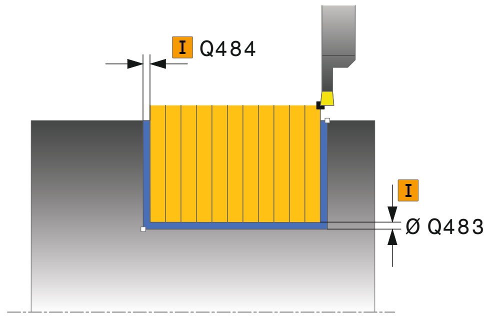
Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the contour starting point
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle between the side at the contour starting point and the perpendicular to the rotary axis



- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of second side** Q496: Angle between the side at the contour end point and the perpendicular to the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Limit to depth** Q463: Max. recess depth per cut



NC blocks

11 CYCL DEF 862 RADIAL RECESSING EXTENDED	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT END OF CONTOUR
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF SECOND SIDE
Q503=+1	;TYPE OF END ELEMENT
Q504=+0.5	;SIZE OF END ELEMENT
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+0	;LIMIT TO DEPTH
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

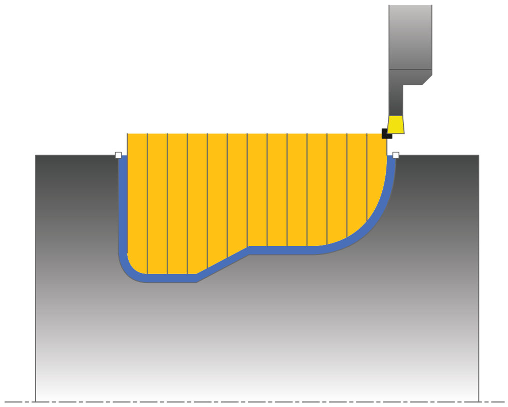
12.24 RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)

Application

This cycle enables you to radially cut in slots of any form.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.



Roughing cycle run

1 The TNC positions the tool at rapid traverse in the Z coordinate (first cut-in position).
2 The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
3 The TNC cuts the area between the starting position and the end point in radial direction at the defined feed rate **Q478**.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (2 to 4) until the slot form is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes one half of the slot at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC finishes the other half of the slot at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

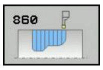
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

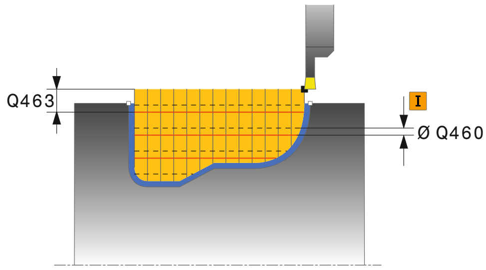
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

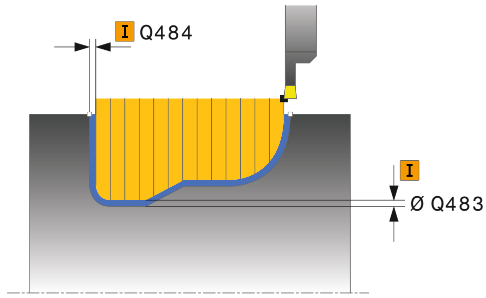
Cycle parameters





- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Limit to depth** Q463: Max. recess depth per cut



NC blocks

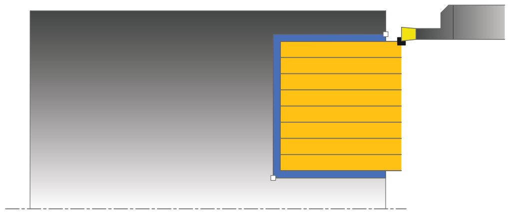
9 CYCL DEF 14.0 CONTOUR	
10 CYCL DEF 14.1 CONTOUR LABEL2	
11 CYCL DEF 860 RECESSING CONTOUR RADIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q479=+0	;CUTTING LIMIT
Q480=+0	;LIMIT VALUE FOR DIAMETER
Q482=+0	;LIMIT VALUE IN Z
Q463=+0	;LIMIT TO DEPTH
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	
14 M30	
15 LBL 2	
16 L X+60 Z-20	
17 L X+45	
18 RND R2	
19 L X+40 Z-25	
20 L Z+0	
21 LBL 0	

12.25 AXIAL RECESSING (Cycle 871, DIN/ISO: G871)

Application

This cycle enables you to axially cut in right-angled slots (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. The cycle processes only the area from the cycle starting point to the end point defined in the cycle.

1. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2. The TNC cuts the area between the starting position and the end point in radial direction at the defined feed rate **Q478**.
3. The TNC positions the tool back at rapid traverse to the beginning of cut.
4. The TNC repeats this process (1 to 3) until the slot width is reached.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

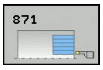
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes half the slot width at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC finishes half the slot width at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

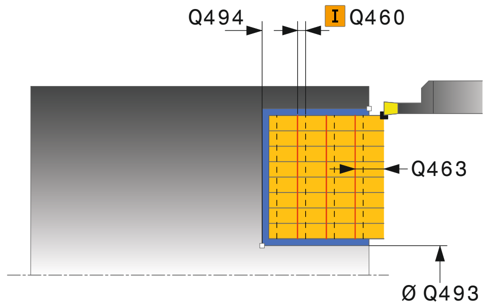
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

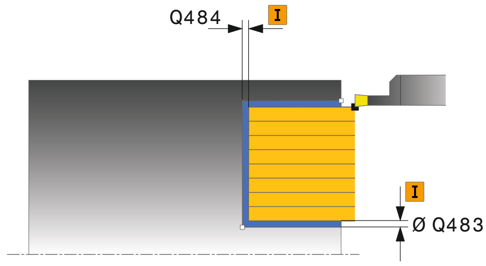
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Limit to depth** Q463: Max. recess depth per cut



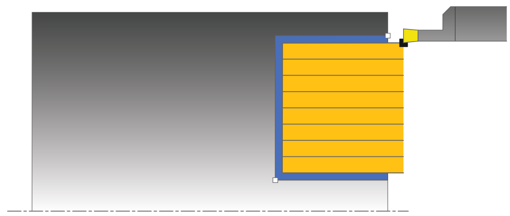


NC blocks

11 CYCL DEF 871 AXIAL RECESSING	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT END OF CONTOUR
Q494=-10	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+0	;LIMIT TO DEPTH
12 L X+65 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

12.26 AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)

Application



This cycle enables you to axially cut in slots (face recessing). Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q492 CONTOUR START IN Z**, the TNC positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2. The TNC cuts the area between the starting position and the end point in radial direction at the defined feed rate **Q478**.
3. The TNC positions the tool back at rapid traverse to the beginning of cut.
4. The TNC repeats this process (1 to 3) until the slot width is reached.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than **Q492 CONTOUR START IN Z**, the TNC positions the tool in the Z coordinate to **Q492** and begins the cycle there.

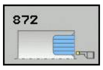
1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3 The TNC returns the tool at rapid traverse.
4 The TNC positions the tool at rapid traverse to the second slot side.
5 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
6 The TNC finishes one half of the slot at the defined feed rate.
7 The TNC positions the tool at rapid traverse to the first side.
8 The TNC finishes the other half of the slot at the defined feed rate.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

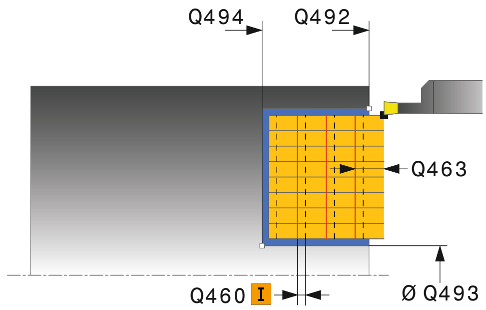
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

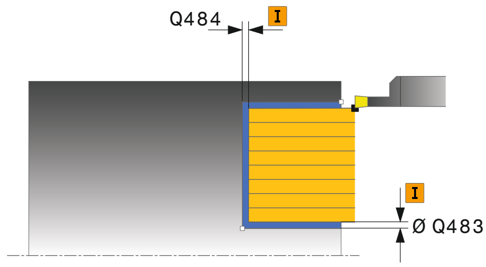
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the contour starting point
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Angle of side** Q495: Angle between the side at the contour starting point and the parallel line to the rotary axis
- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of second side** Q496: Angle between the side at the contour end point and the parallel line to the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius





NC blocks

11 CYCL DEF 871 AXIAL RECESSING EXTENDED	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT END OF CONTOUR
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT

- **Size of end element** Q504: Size of the end element (chamfer section)
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Limit to depth** Q463: Max. recess depth per cut

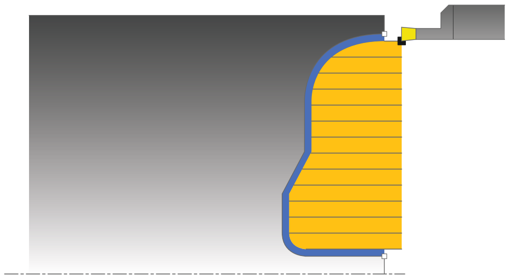
```
Q500=+1.5 ;RADIUS OF CONTOUR EDGE
Q496=+5   ;ANGLE OF SECOND SIDE
Q503=+1   ;TYPE OF END ELEMENT
Q504=+0.5 ;SIZE OF END ELEMENT
Q478=+0.3 ;ROUGHING FEED RATE
Q483=+0.4 ;OVERSIZE FOR DIAMETER
Q484=+0.2 ;OVERSIZE IN Z
Q505=+0.2 ;FINISHING FEED RATE
Q463=+0   ;LIMIT TO DEPTH
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.27 AXIAL RECESSING (Cycle 870, DIN/ISO: G870)

Application

This cycle enables you to axially cut in slots of any form (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.



Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The TNC positions the tool at rapid traverse in the X coordinate (first cut-in position).
2 The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
3 The TNC cuts the area between the starting position and the end point in axial direction at the defined feed rate **Q478**.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (2 to 4) until the slot form is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.

1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3 The TNC finishes one half of the slot at the defined feed rate.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot side.
6 The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7 The TNC finishes the other half of the slot at the defined feed rate.
8 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

!

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

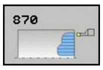
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The tool position at cycle call defines the size of the area to be machined (cycle starting point).

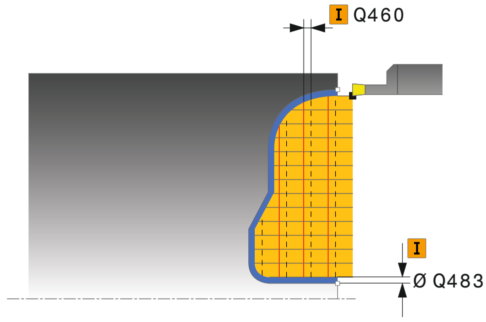
Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

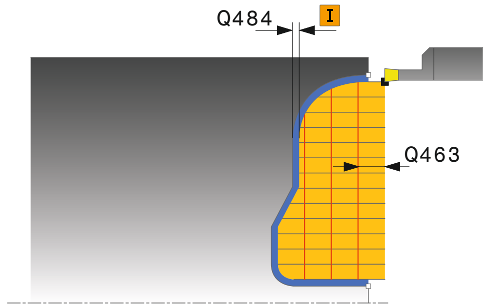
When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Limit to depth** Q463: Max. recess depth per cut





NC blocks

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 870 AXIAL RECESSING
    Q215=+0    ;MACHINING
                OPERATION
    Q460=+2    ;SAFETY CLEARANCE
    Q478=+0.3 ;ROUGHING FEED RATE
    Q483=+0.4 ;OVERSIZE FOR
                DIAMETER
    Q484=+0.2 ;OVERSIZE IN Z
    Q505=+0.2 ;FINISHING FEED RATE
    Q479=+0    ;CUTTING LIMIT
    Q480=+0    ;LIMIT VALUE FOR
                DIAMETER
    Q482=+0    ;LIMIT VALUE IN Z
    Q463=+0    ;LIMIT TO DEPTH
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-15
20 L Z+0
21 LBL 0
```

12.28 THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)

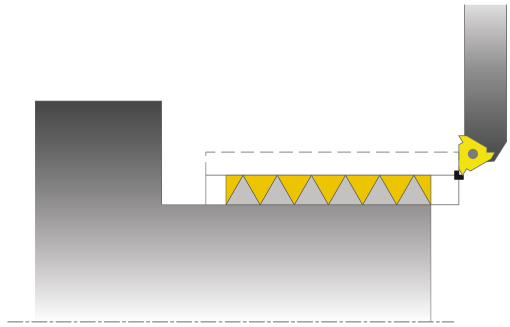
Application

This cycle enables you to run longitudinal turning of threads.

You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth, the cycle uses thread depth in accordance with the ISO1502 standard.

The cycle can be used for inside and outside machining.



Cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.

1 The TNC positions the tool in rapid traverse at set-up clearance in front of the thread and runs an infeed motion.
2 The TNC runs a paraxial longitudinal cut. Here the TNC synchronizes feed rate and speed so that the defined pitch is machined.
3 The TNC retracts the tool at rapid traverse by the set-up clearance.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC runs an infeed motion. The infeeds are run according to the angle of infeed **Q467**.
6 The TNC repeats the process (2 to 5) until the thread depth is completed.
7 The TNC runs the number of air cuts as defined in **Q476**.
8 The TNC repeats the process (2 to 7) according to the number of traverses **Q475**.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The TNC uses the set-up clearance **Q460** as approach path. The approach path must be long enough for the feed axes to be accelerated to the required velocity.

The TNC uses the thread pitch as overrun path. The overrun path must be long enough to decelerate the feed axes.

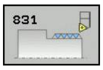
Parameters are available for approach and overrun in Cycle 832 THREAD EXTENDED.

When the TNC runs a thread cut, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

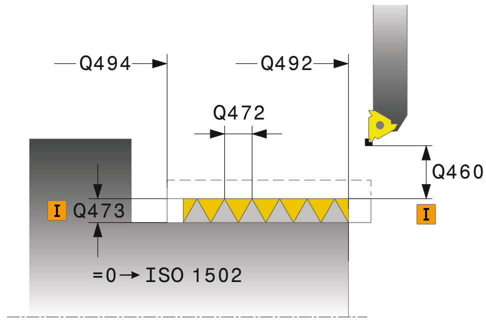
With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use an external tool for inside machining, you can execute machining in the negative diameter range (-X) and reverse the direction of workpiece rotation. Note that with pre-positioning in the negative diameter range, the TNC reverses the effect of the parameter G471 Thread position (external thread is then 1 and internal thread 0).

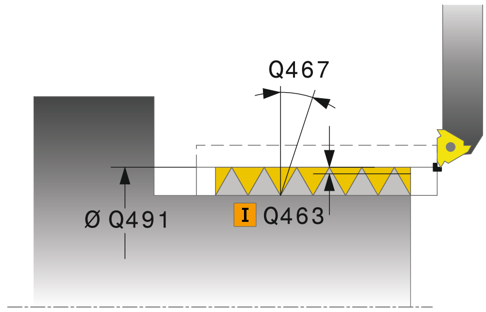
The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.

Cycle parameters



- **Thread position** Q471: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Set-up clearance** Q460: Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) to the synchronized feed rate.
- **Thread diameter** Q491: Define the nominal diameter of the thread.
- **Thread pitch** Q472: Pitch of the thread.
- **Depth of thread** Q473 (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Contour start in Z** Q492: Z coordinate of the starting point
- **Contour end in Z** Q494: Z coordinate of the end point including the runout of the thread Q474.
- **Runout of thread** Q474 (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.
- **Maximum cutting depth** Q463: Maximum plunging depth in radial direction relative to the radius.
- **Angle of infeed** Q467: Angle for the infeed Q463. The reference angle is formed by the perpendicular to the rotary axis.
- **Type of infeed** Q468: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Starting angle** Q470: Angle of the turning spindle at which the thread start is to be made.
- **Number of starts** Q475: Number of thread starts
- **Number of air cuts** Q476: Number of air cuts without infeed at finished thread depth



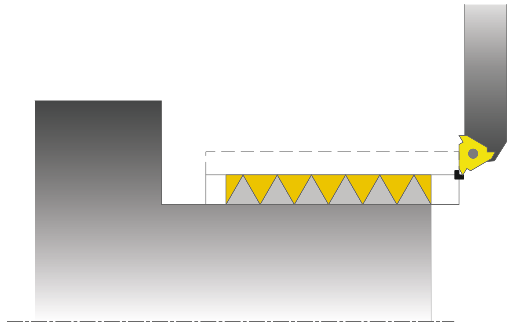


NC blocks

```
11 CYCL DEF 831 LONGITUDINAL
   THREAD
   Q471=+0    ;THREAD POSITION
   Q460=+5    ;SET-UP CLEARANCE
   Q491=+75   ;THREAD DIAMETER
   Q472=+2    ;THREAD PITCH
   Q473=+0    ;THREAD DEPTH
   Q492=+0    ;CONTOUR START IN Z
   Q494=-15   ;CONTOUR END IN Z
   Q474=+0    ;RUNOUT OF THREAD
   Q463=+0.5  ;MAX. CUTTING DEPTH
   Q467=+30   ;ANGLE OF INFEED
   Q468=+0    ;TYPE OF INFEED
   Q470=+0    ;START ANGLE
   Q475=+30   ;NUMBER OF STARTS
   Q476=+30   ;NO. OF AIR CUTS
12 L X+80 Y+0 Z+2 FMAX M303
13 CYCL CALL
```

12.29 THREAD EXTENDED (Cycle 832, DIN/ISO: G832)

Application



This cycle enables you to run both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- Selection of longitudinal thread or face thread.
- The parameters for dimension type of taper, taper angle and contour starting point X enable the definition of various tapered threads.
- The parameters for approach path and overrun path define a path in which feed axes can be accelerated or decelerated.

You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.

1 The TNC positions the tool in rapid traverse at set-up clearance in front of the thread and runs an infeed motion.
2 The TNC runs a longitudinal cut. Here the TNC synchronizes feed rate and speed so that the defined pitch is machined.
3 The TNC retracts the tool at rapid traverse by the set-up clearance.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC runs an infeed motion. The infeeds are run according to the angle of infeed **Q467**.
6 The TNC repeats the process (2 to 5) until the thread depth is completed.
7 The TNC runs the number of air cuts as defined in **Q476**.
8 The TNC repeats the process (2 to 7) according to the number of traverses **Q475**.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.

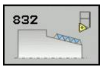
The overrun path (**Q466**) must be long enough to decelerate the feed axes.

When the TNC runs a thread cut, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

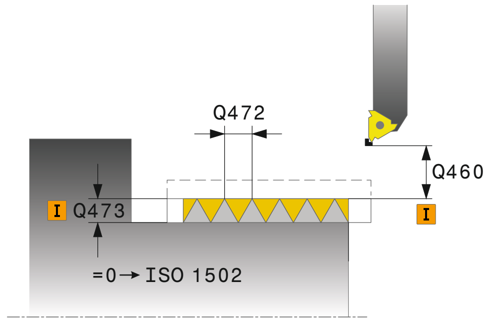
With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use an external tool for inside machining, you can execute machining in the negative diameter range (-X) and reverse the direction of workpiece rotation. Note that with pre-positioning in the negative diameter range, the TNC reverses the effect of the parameter G471 Thread position (external thread is then 1 and internal thread 0).

The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.

Cycle parameters



- **Thread position** Q471: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Thread orientation**Q461: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)
- **Set-up clearance** Q460: Set-up clearance perpendicular to thread pitch.
- **Thread pitch** Q472: Pitch of the thread.
- **Depth of thread** Q473 (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Dimension type of taper** Q464: Define the type of dimension for the taper contour:
  0: Via starting point and end point
  1: Via end point, start-X and taper angle
  2: Via end point, start-Z and taper angle
  3: Via starting point, end-X and taper angle
  4: Via starting point, end-Z and taper angle
- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)
- **Contour start in Z** Q492: Z coordinate of the starting point
- **Diameter at end of contour** Q493: X coordinate of the end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the end point
- **Taper angle** Q469: Taper angle of contour
- **Runout of thread** Q474 (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.
- **Approach path** Q465 (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside of the defined thread contour.
- **Overrun path** Q466: Length of the path in pitch direction on which the feed axes are decelerated. The overrun path is within the defined thread contour.
- **Maximum cutting depth** Q463: Maximum plunging depth perpendicular to the thread pitch



NC blocks

11 CYCL DEF 832 THREAD EXTENDED	
Q471=+0	;THREAD POSITION
Q461=+0	;THREAD ORIENTATION
Q460=+2	;SET-UP CLEARANCE
Q472=+2	;THREAD PITCH
Q473=+0	;THREAD DEPTH
Q464=+0	;DIMENSION TYPE FOR TAPER
Q491=+100	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+110	;DIAMETER AT END OF CONTOUR
Q494=-35	;CONTOUR END IN Z
Q469=+0	;TAPER ANGLE
Q474=+0	;RUNOUT OF THREAD
Q465=+4	;APPROACH PATH
Q466=+4	;OVERRUN PATH
Q463=+0.5	;MAX. CUTTING DEPTH
Q467=+30	;ANGLE OF INFEED
Q468=+0	;TYPE OF INFEED
Q470=+0	;START ANGLE
Q475=+30	;NUMBER OF STARTS
Q476=+30	;NO. OF AIR CUTS
12 L X+80 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

- **Angle of infeed** Q467: Angle for the infeed Q463. The reference angle is formed by the parallel line to the thread pitch.
- **Type of infeed** Q468: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Starting angle** Q470: Angle of the turning spindle at which the thread start is to be made.
- **Number of starts** Q475: Number of thread starts
- **Number of air cuts** Q476: Number of air cuts without infeed at finished thread depth

12.30 CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ISO: G830)

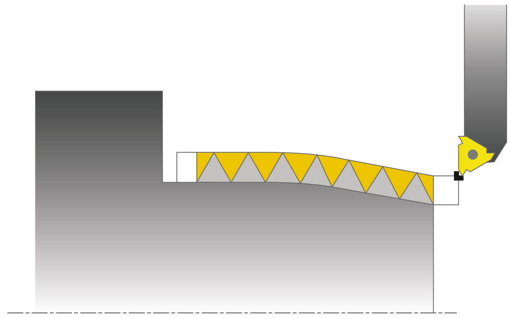
Application

This cycle enables you to run both face turning and longitudinal turning of threads with any form.

You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.



> **!** The cycle 830 runs the overrun **Q466** following the programmed contour. Note the spatial conditions.

Cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.

1 The TNC positions the tool in rapid traverse at set-up clearance in front of the thread and runs an infeed motion.
2 The TNC runs a thread cut parallel to the defined thread contour. Here the TNC synchronizes feed rate and speed so that the defined pitch is machined.
3 The TNC retracts the tool at rapid traverse by the set-up clearance.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC runs an infeed motion. The infeeds are run according to the angle of infeed **Q467**.
6 The TNC repeats the process (2 to 5) until the thread depth is completed.
7 The TNC runs the number of air cuts as defined in **Q476**.
8 The TNC repeats the process (2 to 7) according to the number of traverses **Q475**.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.

The overrun path (**Q466**) must be long enough to decelerate the feed axes.

Both the approach and overrun take place outside the defined contour.

When the TNC runs a thread cut, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

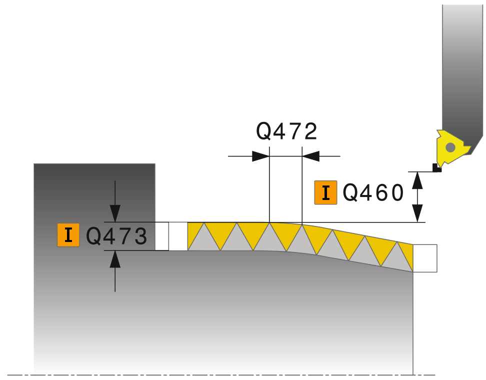
When you use local **QL** Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use an external tool for inside machining, you can execute machining in the negative diameter range (-X) and reverse the direction of workpiece rotation. Note that with pre-positioning in the negative diameter range, the TNC reverses the effect of the parameter G471 Thread position (external thread is then 1 and internal thread 0).

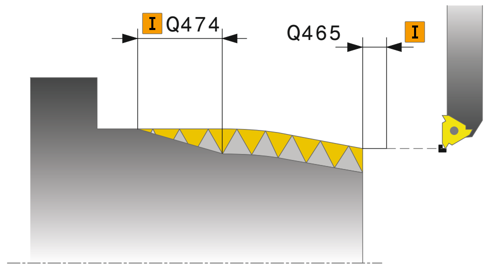
The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.

Cycle parameters

- **Thread position** Q471: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Thread orientation**Q461: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)
- **Set-up clearance** Q460: Set-up clearance perpendicular to thread pitch.
- **Thread pitch** Q472: Pitch of the thread.
- **Depth of thread** Q473 (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Runout of thread** Q474 (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.



- **Approach path** Q465 (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside of the defined thread contour.
- **Overrun path** Q466: Length of the path in pitch direction on which the feed axes are decelerated. The overrun path is within the defined thread contour.
- **Maximum cutting depth** Q463: Maximum plunging depth perpendicular to the thread pitch
- **Angle of infeed** Q467: Angle for the infeed Q463. The reference angle is formed by the parallel line to the thread pitch.
- **Type of infeed** Q468: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Starting angle** Q470: Angle of the turning spindle at which the thread start is to be made.
- **Number of starts** Q475: Number of thread starts
- **Number of air cuts** Q476: Number of air cuts without infeed at finished thread depth



NC blocks

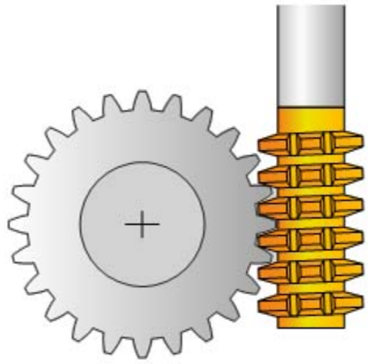
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 830 CONTOUR-PARALLEL THREAD
Q471=+0 ;THREAD POSITION
Q461=+0 ;THREAD ORIENTATION
Q460=+2 ;SET-UP CLEARANCE
Q472=+2 ;THREAD PITCH
Q473=+0 ;THREAD DEPTH
Q474=+0 ;RUNOUT OF THREAD
Q465=+4 ;APPROACH PATH
Q466=+4 ;OVERRUN PATH
Q463=+0.5 ;MAX. CUTTING DEPTH
Q467=+30 ;ANGLE OF INFEED
Q468=+0 ;TYPE OF INFEED
Q470=+0 ;START ANGLE
Q475=+30 ;NUMBER OF STARTS
Q476=+30 ;NO. OF AIR CUTS
12 L X+80 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L X+70 Z-30
18 RND R60
19 L Z-45
20 LBL 0

12.31 GEAR HOBBING (Cycle 880, DIN/ISO: G880)

Cycle run

With Cycle 880 Gear Hobbing you can machine external cylindrical gears or helical gears with any angles. In the cycle you first define the **gear** and then the **tool** with which the gear is to be machined. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary motion of the tool spindle and rotary table. In addition, the gear hob moves along the workpiece in axial direction.

While Cycle 880 Gear Hobbing is active, the coordinate system might be rotated. It is therefore essential to program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145** at the end of the cycle.



Cycle run:

1 The TNC positions the tool in the tool axis to clearance height Q260 at the feed rate FMAX. If the value of the current tool location in the tool axis is greater than Q260, the tool is not moved.
2 Before tilting the working plane, the TNC positions the tool in X to a safe coordinate at the feed rate FMAX. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The TNC then tilts the working plane at the feed rate Q253; **M144** is internally active in the cycle.
4 The TNC positions the tool at the feed rate FMAX to the starting point in the working plane.
5 The TNC then moves the tool in the tool axis at the feed rate Q253 to the set-up clearance Q460.
6 The TNC moves the hob in the longitudinal direction at the programmed feed rate Q478 (for roughing) or Q505 (for finishing) along the workpiece into which the teeth are to be cut. The area to be machined is limited by the starting point in Z Q551+Q460 and the end point in Z Q552+Q460.
7 When the tool reaches the end point, it is retracted at the feed rate Q253 and returns to the starting point.
8 The TNC repeats the steps 5 to 7 until the defined gear is completed.
9 Finally, the TNC retracts the tool to the clearance height Q260 at the feed rate FMAX.
10 The machining operation ends in the tilted system.
11 Now you need to move the tool to a safe height and reset the tilting of the working plane.
12 Then you must program Cycle 801 RESET ROTARY COORDINATE SYSTEM and **M145**.

Please note while programming:

The values entered for module, number of teeth and outside diameter are monitored. If these values are not consistent, an error message is displayed. It is also possible to make entries for only 2 of these 3 parameters. In this case, enter the value 0 for either the module or the number of teeth or the outside diameter. The TNC then calculates the missing value.

Program FUNCTION TURNDATA SPIN VCONST:OFF.

If you program FUNCTION TURNDATA SPIN VCONST:OFF S15, the spindle speed of the tool is calculated as follows: Q541 x S. With Q541=238 and S=15, this would result in a tool spindle speed of 3570 rpm.

Define the tool as a milling cutter in the tool table.

To avoid exceeding the maximum permissible spindle speed of the tool, you can enter a limitation. (Entry in the "Nmax" column of the tool table "tool.t".)

Before starting the cycle, program the direction of rotation of the workpiece (M303/M304).

Before cycle call, set the datum to the center of rotation.

Cycle 880 Gear Hobbing is run in turning mode and is CALL-active.

Software option 50 must be enabled

Danger of collision!

Pre-position the tool so that it is already on the desired machining side Q550. On this machining side, move the tool to a safe position where there is no danger of collision with the workpiece (clamping devices) during tilting.

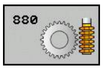
Please note that the starting point in Z and the end point in Z are extended by the set-up clearance Q460! Clamp the workpiece in such a way that there is no danger of collision between the tool and the clamping devices!

If you program M136 before the cycle, the TNC interprets feed rate values in the cycle in mm/rev., and if you are not using M136, in mm/min!

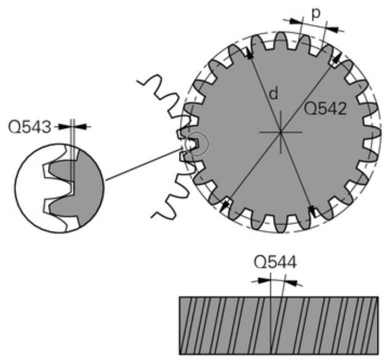
After Cycle 880 GEAR HOBBING remember to call Cycle 801 and M145 in order to reset the coordinate system.

If you abort a program during machining, you must **reset the coordinate system with Cycle 801** and **call M145** before you start machining again!

Cycle parameters



- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Module** Q540: Define the gear: Module of the gear wheel. Input range 0 to 99.9999
- **Number of teeth** Q541: Define the gear: Number of teeth. Input range 0 to 99999
- **Outside diameter** Q542: Define the gear: Outside diameter of the finished part. Input range 0 to 99999.9999
- **Trough-tip clearance** Q543: Define the gear: Distance between the tip circle of the gear to be cut and the root circle of the mating gear. Input range 0 to 9.9999
- **Angle of inclination** Q544: Define the gear: Angle by which the teeth in a helical gear are inclined relative to the axis direction. (With a straight-cut gear, this angle is 0°) Input range -45 to +45
- **Tool lead angle** Q545: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (e.g. 0°47'=0.7833) Input range -60.0000 to +60.0000
- **Change tool direction (3, 4)** Q546: Define the tool: Direction of spindle rotation of the gear hob:
  3: Right-turning tool (M3)
  4: Left-turning tool (M4)
- **Ang. offset, spindle** Q547: Angle by which the TNC rotates the workpiece at cycle start. Input range -180.0000 to +180.0000
- **Machining side** Q550: Define the side on which the machining operation is to be performed.
  0: Positive machining side
  1: Negative machining side
- **Preferred direction** Q533: Selection of alternate possibilities of inclination.
  0: Solution using the shortest path
  -1: Solution in the negative direction
  +1: Solution in the positive direction
- **Inclined machining** Q530: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically, thereby orienting the tool tip (MOVE). The relative position between the tool and workpiece remains unchanged. The TNC performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN).



NC blocks

63 CYCL DEF 880 GEAR HOBBING	
Q215=0	;MACHINING OPERATION
Q540=0	;MODULE
Q541=0	;NUMBER OF TEETH
Q542=0	;OUTSIDE DIAMETER
Q543=0.167	;TROUGH-TIP CLEARANCE
Q544=0	;ANGLE OF INCLINATION
Q545=0	;TOOL LEAD ANGLE
Q546=3	;CHANGE TOOL DIRECTN.
Q547=0	;ANG. OFFSET, SPINDLE
Q550=1	;MACHINING SIDE
Q533=0	;PREFERRED DIRECTION
Q530=2	;INCLINED MACHINING
Q253=750	;F PRE-POSITIONING
Q260=100	;CLEARANCE HEIGHT
Q553=10	;TOOL LENGTH OFFSET
Q551=0	;STARTING POINT IN Z
Q552=-10	;END POINT IN Z
Q463=1	;MAX. CUTTING DEPTH
Q460=2	;SET-UP CLEARANCE
Q488=0.3	;PLUNGING FEED RATE
Q478=0.3	;ROUGHING FEED RATE
Q483=0.4	;OVERSIZE FOR DIAMETER
Q505=0.2	;FINISHING FEED RATE

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Entry in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the tool axis at which no collision between tool and workpiece (fixtures) can occur. Default value: **PREDEF**, input range -99999.9999 to 99999.9999
- **Tool length offset** Q553: Define which section of the gear hob is to be used. As gear hobbing causes wear to the hob teeth, the tool can be offset in the longitudinal direction to evenly apply the load over the entire length of the tool. In parameter Q553 you enter an incremental distance by which the tool is to be moved in the longitudinal direction. Input range 0 to 99.9999
- **Starting point in Z** Q551: Starting point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- **End point in Z** Q552: End point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.Input range 0 to 999.999
- **Feed rate for plunging** Q488: Feed rate for tool infeed. Input range 0 to 99999.999
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour .
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

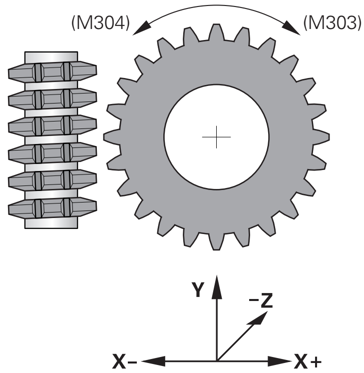
Direction of rotation depending on the machining side (Q550)

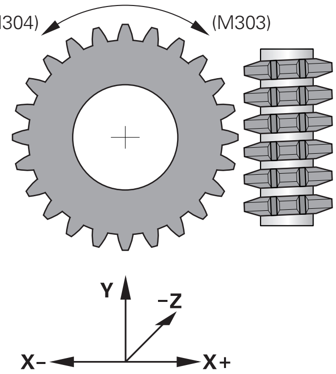
Determine the direction of rotation of the rotary table:

1. **What tool? (Right-cutting / left-cutting)?**
2. **What machining side? X+ (Q550=0) / X- (Q550=1)**
3. **Look up the direction of rotation of the rotary table in one of the two tables below!** Select the table for the direction of rotation of your tool (**right-cutting/left-cutting**). In this table, look up the direction of rotation of the rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3	
Machining side X+ (Q550=0)	Direction of table rotation: clockwise (M303)
Machining side X- (Q550=1)	Direction of table rotation: counterclockwise (M304)

Tool: Left-cutting M4	
Machining side X+ (Q550=0)	Direction of table rotation: counterclockwise (M304)
Machining side X- (Q550=1)	Direction of table rotation: clockwise (M303)

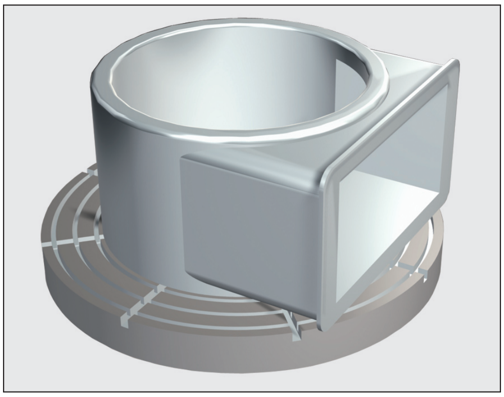




12.32 CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)

Application

When turning a nonsymmetrical workpiece, such as a pump body, an unbalance may occur. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle 892 CHECK UNBALANCE the TNC checks the unbalance of the turning spindle. This cycle uses two parameters. Q450 describes the maximum unbalance and Q451 the maximum speed. **If the maximum unbalance is exceeded, an error message is displayed and the program is aborted.** When the maximum unbalance is not exceeded, the TNC executes the program without interruptions. This function protects the machine mechanics. It enables you to take action if excessive unbalance is detected.



Please note while programming:

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.

Software option 50 must be enabled

This function is executed in turning mode. FUNCTION MODE TURN must be active, otherwise the TNC generates an error message.

Your machine tool builder configures Cycle 892.

Your machine tool builder defines the function of Cycle 892.

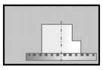
The turning spindle rotates during the unbalance check.

This function can also be run on machines with more than one turning spindle. Contact the machine tool builder for further information.

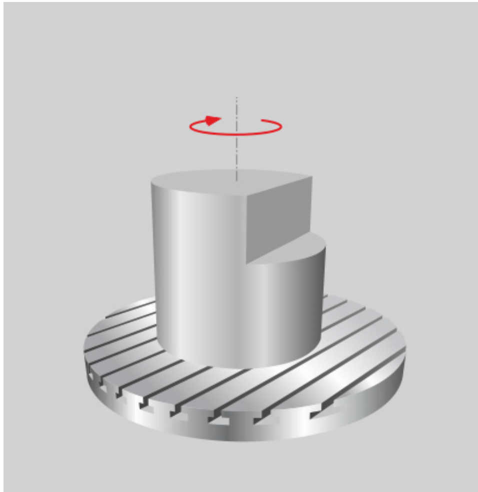
You need to check the applicability of the control's internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.

If Cycle 892 CHECK UNBALANCE has aborted a program, it is recommended to use the manual MEASURE UNBALANCE cycle. With this cycle, the TNC determines the unbalance and calculates the mass and position of a balancing weight. For more information on the manual MEASURE UNBALANCE cycle, refer to the User's Manual for Conversational Programming.

Cycle parameters



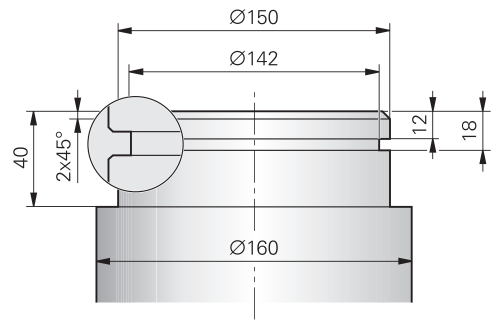
- **Maximum runout** Q450: (mm) Specifies the maximum amplitude of a sinusoidal unbalance signal. The signal results from the following error of the measuring axis and from the spindle revolutions.
- **Speed** Q451: (rpm) The unbalance check starts at a low rotational speed (e.g. 50 rpm). It is automatically increased by specified increments (e.g. 25 rpm) until the defined maximum speed is reached. Spindle override is not effective.



```
63
CYCL DEF 892 CHECK UNBALANCE
    Q450=0      ;MAXIMUM RUNOUT
    Q451=50     ;SPEED
```

12.33 Example program

Example: Shoulder with recess



0 BEGIN PGM SHOULDER MM	
1 BLK FORM 0.1 Y X+0 Y-10 Z-35	Definition of workpiece blank
2 BLK FORM 0.2 X+87 Y+10 Z+2	
3 TOOL CALL 12	Tool call
4 M140 MB MAX	Retract the tool
5 FUNCTION MODE TURN	Activate Turning mode
6 FUNCTION TURNDATA SPIN VCONST:ON VC:150	Constant surface speed
7 CYCL DEF 800 ADAPT ROTARY COORDINATE SYSTEM	Cycle definition adapt rotary coordinate system
Q497=+0 ;PRECISION ANGLE	
Q498=+0 ;REVERSE TOOL	
8 M136	Feed rate in mm per revolution
9 L X+165 Y+0 R0 FMAX	Move to starting point in the plane
10 L Z+2 R0 FMAX M304	Set-up clearance, turning spindle on
11 CYCL DEF 812 SHOULDER LONG. EXTENDED.	Cycle definition shoulder longitudinal
Q215=+0 ;MACHINING OPERATION	
Q460=+2 ;SAFETY CLEARANCE	
Q491=+160 ;DIAMETER AT CONTOUR START	
Q492=+0 ;CONTOUR START IN Z	
Q493=+150 ;DIAMETER AT END OF CONTOUR	
Q494=-40 ;CONTOUR END IN Z	
Q495=+0 ;ANGLE OF CIRCUM. SURFACE	
Q501=+1 ;TYPE OF STARTING ELEMENT	
Q502=+2 ;SIZE OF STARTING ELEMENT	
Q500=+1 ;RADIUS OF CONTOUR EDGE	
Q496=+0 ;ANGLE OF FACE	
Q503=+1 ;TYPE OF END ELEMENT	
Q504=+2 ;SIZE OF END ELEMENT	
Q463=+2.5 ;MAX. CUTTING DEPTH	
Q478=+0.25 ;ROUGHING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	

Q484=+0.2 ;OVERSIZE IN Z	
Q505=+0.2 ;FINISHING FEED RATE	
Q506=+0 ;CONTOUR SMOOTHING	
12 CYCL CALL M8	Cycle call
13 M305	Turning spindle off
14 TOOL CALL 15	Tool call
15 M140 MB MAX	Retract the tool
16 FUNCTION TURNDATA SPIN VCONST:ON VC:100	Constant cutting speed
17 CYCL DEF 800 ADAPT ROTARY COORDINATE SYSTEM	Cycle definition adapt rotary coordinate system
Q497=+0 ;PRECISION ANGLE	
Q498=+0 ;REVERSE TOOL	
18 L X+165 Y+0 R0 FMAX	Move to starting point in the plane
19 L Z+2 R0 FMAX M304	Set-up clearance, turning spindle on
20 CYCL DEF 862 RADIAL RECESSING EXTENDED	Cycle definition recess
Q215=+0 ;MACHINING OPERATION	
Q460=+2 ;SAFETY CLEARANCE	
Q491=+150 ;DIAMETER AT CONTOUR START	
Q492=-12 ;CONTOUR START IN Z	
Q493=+142 ;DIAMETER AT END OF CONTOUR	
Q494=-18 ;CONTOUR END IN Z	
Q495=+0 ;ANGLE OF SIDE	
Q501=+1 ;TYPE OF STARTING ELEMENT	
Q502=+1 ;SIZE OF STARTING ELEMENT	
Q500=+0 ;RADIUS OF CONTOUR EDGE	
Q496=+0 ;ANGLE OF SECOND SIDE	
Q503=+1 ;TYPE OF END ELEMENT	
Q504=+1 ;SIZE OF END ELEMENT	
Q478=+0.3 ;ROUGHING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	
Q484=+0.2 ;OVERSIZE IN Z	
Q505=+0.15 ;FINISHING FEED RATE	
Q463=+0 ;LIMIT TO DEPTH	
21 CYCL CALL M8	Cycle call
22 M305	Turning spindle off
23 M137	Feed rate in mm per minute
24 M140 MB MAX	Retract the tool
25 FUNCTION MODE MILL	Activate Milling mode
26 M30	End of program
27 END PGM SHOULDER MM	

Example: Gear hobbing

Cycle 880 GEAR HOBBING is used in the following program. This programming example illustrates the machining of a helical gear, with Module=2.1.

Program sequence

- Tool call: Gear hob
- Start turning mode
- Approach safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R42 L150	Definition of workpiece blank: Cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TOOL CALL "GEAR_HOB_D75"	Call the tool
4 FUNCTION MODE TURN	Activate turning mode
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	Reset the coordinate system
4 M145	Deactivate M144 if still active
4 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
4 M140 MB MAX	Retract the tool
4 L A+0 R0 FMAX	Set the rotary axis to 0
4 L X+250 Y-250 R0 FMAX	Pre-position the tool in the working plane on the side on which machining will be performed
4 Z+20 R0 FMAX	Pre-position the tool in the spindle axis
4 L M136	Feed rate in mm/rev
5 CYCL DEF 880 GEAR HOBBING	Activate interpolation turning
Q215=+0 ;MACHINING OPERATION	
Q540=+2.1 ;MODULE	
Q541=+0 ;NUMBER OF TEETH	
Q542=+69.3 ;OUTSIDE DIAMETER	
Q543=+0.1666 ;TROUGH-TIP CLEARANCE	
Q544=-5 ;ANGLE OF INCLINATION	
Q545=+1.6833 ;TOOL LEAD ANGLE	
Q546=+3 ;CHANGE TOOL DIRECTN.	
Q550=+0 ;MACHINING SIDE	
Q533=+0 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	
Q253=+2000 ;F PRE-POSITIONING	
Q260=+20 ;CLEARANCE HEIGHT	
Q553=+10 ;TOOL LENGTH OFFSET	
Q551=+0 ;STARTING POINT IN Z	
Q552=-10 ;END POINT IN Z	
Q463=+1 ;MAX. CUTTING DEPTH	

Q488=+1 ;PLUNGING FEED RATE	
Q478=+2 ;ROUGHING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	
Q505=+1 ;FINISHING FEED RATE	
6 CYCL CALL M303	Call the cycle, spindle on
7 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	Reset the coordinate system
8 M145	Deactivate the active M144 in the cycle
9 FUNCTION MODE MILL	Activate milling mode
10 M140 MB MAX	Retract the tool in the tool axis
11 L A+0 C+0 R0 FMAX	Reset the rotation
12 M30	END of program
44 END PGM 5 MM	

13

Using Touch Probe Cycles

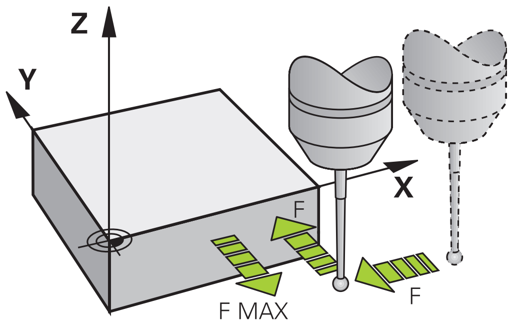
13.1 General information about touch probe cycles

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.

Method of function

Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder determines the probing feed rate in a machine parameter (see "Before You Start Working with Touch Probe Cycles" later in this chapter).



When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the TNC: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the TNC displays an error message (distance: **DIST** from touch probe table).

Consideration of a basic rotation in the Manual Operation mode

During probing the TNC considers an active basic rotation and approaches the workpiece at an angle.

Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

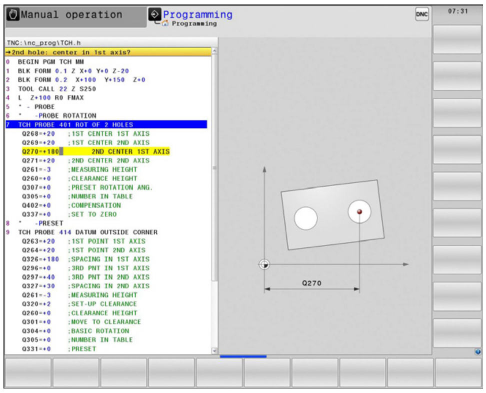
In the **Manual Operation** and **El. Handwheel** modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensating workpiece misalignment
- Setting datums

Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual and El. Handwheel modes, the TNC provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Datum setting
- Automatic workpiece inspection
- Automatic tool measurement



You can program the touch probe cycles in the Programming and Editing operating mode via the TOUCH PROBE key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the TNC shows a graphic during cycle definition. The graphic shows the parameter that needs to be entered (see figure at right).

Defining the touch probe cycle in the Programming and Editing mode of operation

TOUCH PROBE

- The soft-key row shows all available touch probe functions divided into groups.
- Select the desired probe cycle group, for example datum setting. Cycles for automatic tool measurement are available only if your machine has been prepared for them.
- Select a cycle, e.g. datum setting at pocket center. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the ENT key.
- The TNC ends the dialog when all required data has been entered

Group of measuring cycles	Soft key	Page
Cycles for automatic measurement and compensation of workpiece misalignment		448
Cycles for automatic workpiece presetting		470
Cycles for automatic workpiece inspection		522
Special cycles	SPECIAL CYCLES	564
Cycles for automatic tool measurement (enabled by the machine tool builder)		612

NC blocks

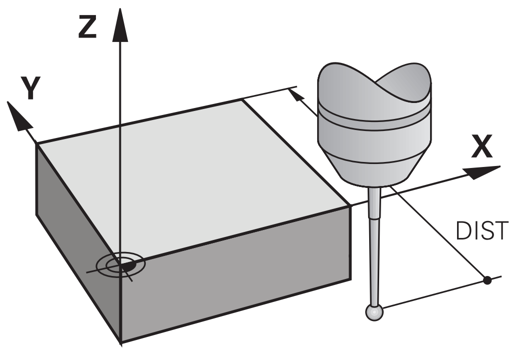
```
5 TCH PROBE 410 DATUM INSIDE
   RECTAN.
   Q321=+50  ;CENTER IN 1ST AXIS
   Q322=+50  ;CENTER IN 2ND AXIS
   Q323=60   ;FIRST SIDE LENGTH
   Q324=20   ;2ND SIDE LENGTH
   Q261=-5   ;MEASURING HEIGHT
   Q320=0    ;SET-UP CLEARANCE
   Q260=+20  ;CLEARANCE HEIGHT
   Q301=0    ;MOVE TO CLEARANCE
   Q305=10   ;NO. IN TABLE
   Q331=+0   ;DATUM
   Q332=+0   ;DATUM
   Q303=+1   ;MEAS. VALUE
              TRANSFER
   Q381=1    ;PROBE IN TS AXIS
   Q382=+85  ;1ST CO. FOR TS AXIS
   Q383=+50  ;2ND CO. FOR TS AXIS
   Q384=+0   ;3RD CO. FOR TS AXIS
   Q333=+0   ;DATUM
```

13.2 Before You Start Working with Touch Probe Cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

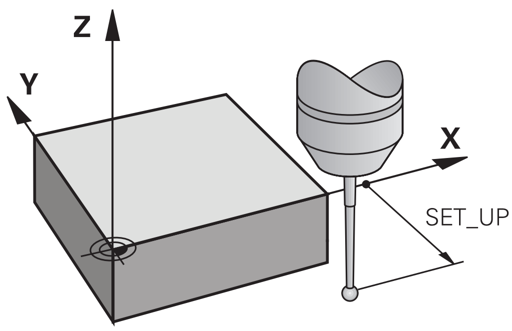
Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the path defined in **DIST**, the TNC outputs an error message.



Set-up clearance to touch point: SET_UP in touch probe table

In **SET_UP** you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to **SET_UP**.



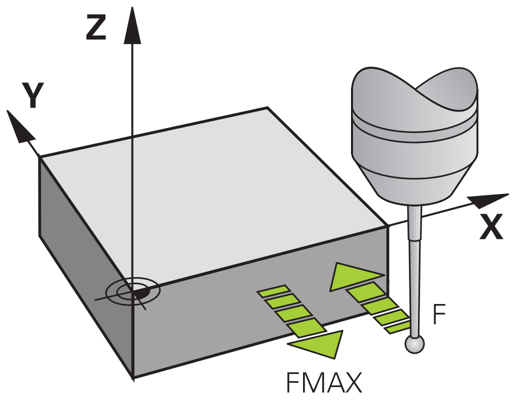
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: F in touch probe table

In **F** you define the feed rate at which the TNC is to probe the workpiece.



Touch trigger probe, rapid traverse for positioning: FMAX

In **FMAX** you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table

In **F_PREPOS** you define whether the TNC is to position the touch probe at the feed rate defined in FMAX or at rapid traverse.

- Input value = **FMAX_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX_MACHINE**: Pre-position at rapid traverse

Multiple measurements

To increase measuring certainty, the TNC can run each probing process up to three times in sequence. Define the number of measurements in machine parameter **ProbeSettings** > **Configuration of probe behavior** > **Automatic mode: Multiple measurements with probe function**. If the measured position values differ too greatly, the TNC outputs an error message (the limit value is defined in **Confidence interval of multiple measurements**). With multiple measurement it is possible to detect random errors, e.g. from contamination.

If the measured values lie within the confidence interval, the TNC saves the mean value of the measured positions.

Confidence interval of multiple measurements

When you perform a multiple measurement, you store the value that the measured values may vary in **ProbeSettings** > **Configuration of probe behavior** > **Automatic mode: Confidence interval of multiple measurements**. If the difference in the measured values exceeds the value defined by you, the TNC outputs an error message.

Executing touch probe cycles

All touch probe cycles are DEF active. This means that the TNC runs the cycle automatically as soon as the TNC executes the cycle definition in the program run.

Danger of collision!
When running touch probe cycles, no cycles must be active for coordinate transformation (Cycle 7 DATUM, Cycle 8 MIRROR IMAGE, Cycle 10 ROTATION, Cycles 11 SCALING and 26 AXIS-SPECIFIC SCALING).

You can also run the Touch Probe Cycles 408 to 419 during an active basic rotation. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT with datum tables after the measuring cycle.

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (defined in the cycle), the TNC retracts the touch probe in the probe axis to the clearance height and then positions it in the working plane to the first starting position.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the TNC first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height.

13.3 Touch probe table

General information

Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you run several touch probes on your machine tool, you can save separate data for each touch probe.

Editing touch probe tables

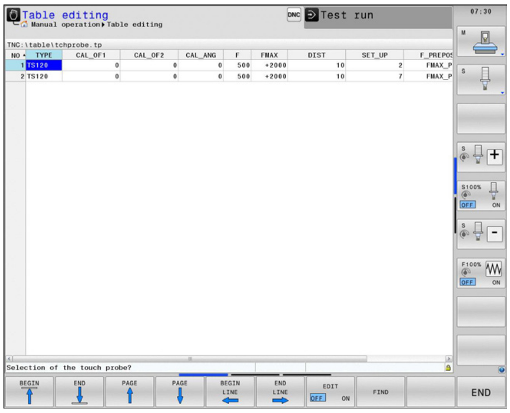
To edit the touch probe table, proceed as follows:

- Select the **Manual Operation** mode

- Select the touch probe functions: Press the **TOUCH PROBE** soft key The TNC displays additional soft keys

- Select the touch probe table: Press the **TCH PROBE TABLE** soft key

- Set the **EDIT** soft key to **ON**
- Using the arrow keys, select the desired setting.
- Perform desired changes.
- Exit the touch probe table: Press the **END** soft key



Touch probe data

Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool number	–
TYPE	Selection of the touch probe used	**Selection of touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis for the reference axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis for the minor axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	The TNC orients the touch probe to the orientation angle before calibration or probing (if orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the TNC is to probe the workpiece	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe pre-positions, or is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within the defined path, the TNC outputs an error message	**Maximum measuring path? [mm]**
SET_UP	In **SET_UP** you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to the **SET_UP** machine parameter.	**Set-up clearance? [mm]**
F_PREPOS	Defining speed with pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-positioning at rap. traverse? ENT/NO ENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Orient touch probe cycles? Yes=ENT, No=NOENT**

14

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.1 Fundamentals

Overview

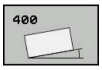
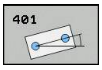
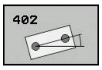
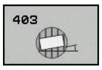
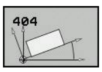
When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

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.

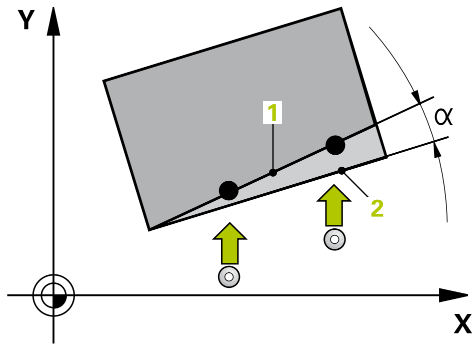
Refer to your machine manual.

The TNC provides five cycles that enable you to measure and compensate workpiece misalignment. In addition, you can reset a basic rotation with Cycle 404:

Cycle	Soft key	Page
400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.		450
401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.		453
402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.		456
403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.		459
405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.		463
404 SET BASIC ROTATION Setting any basic rotation.		462

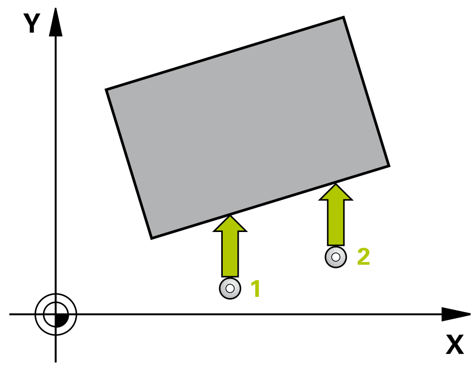
Characteristics common to all touch probe cycles for measuring workpiece misalignment

For Cycles 400, 401 and 402 you can define through parameter Q307 **Default setting for basic rotation** whether the measurement result is to be corrected by a known angle # (see figure at right). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.



14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

Cycle run



Touch probe cycle 400 determines a workpiece misalignment by measuring two points, which must lie on a straight surface. With the basic rotation function the TNC compensates the measured value.

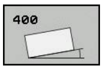
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the programmed touch point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves to the next starting position **2** and probes the second position.
4 The TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

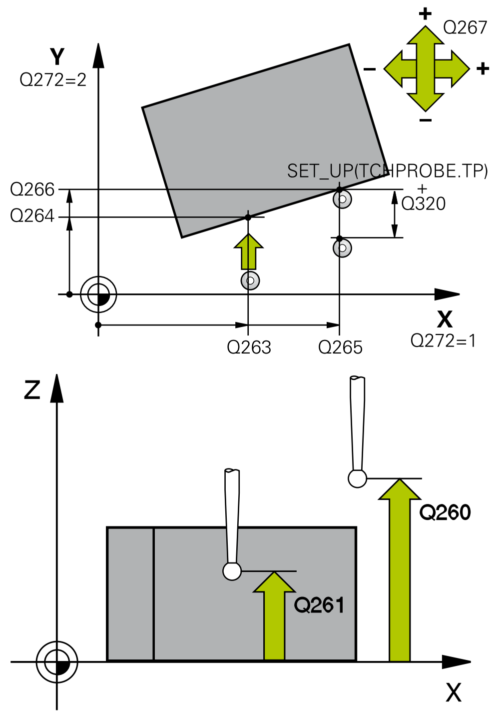
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC will reset an active basic rotation at the beginning of the cycle.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative Traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999



NC blocks

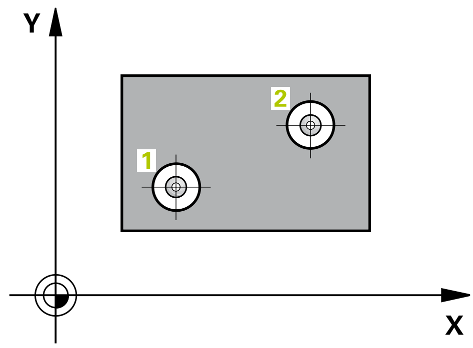
5 TCH PROBE 400 BASIC ROTATION	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+3.5	;1ST POINT 2ND AXIS
Q265=+25	;2ND POINT 1ST AXIS
Q266=+2	;2ND POINT 2ND AXIS
Q272=2	;MEASURING AXIS
Q267=+1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q307=0	;PRESET ROT. ANGLE
Q305=0	;NO. IN TABLE

- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Preset value for rotation angle** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. Input range 0 to 99999

14.3 BASIC ROTATION over two holes (Cycle 401, DIN/ISO: G401)

Cycle run

The Touch Probe Cycle 401 measures the centers of two holes. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the hole centers. With the basic rotation function, the TNC compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the center of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5 Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

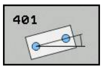
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC will reset an active basic rotation at the beginning of the cycle.

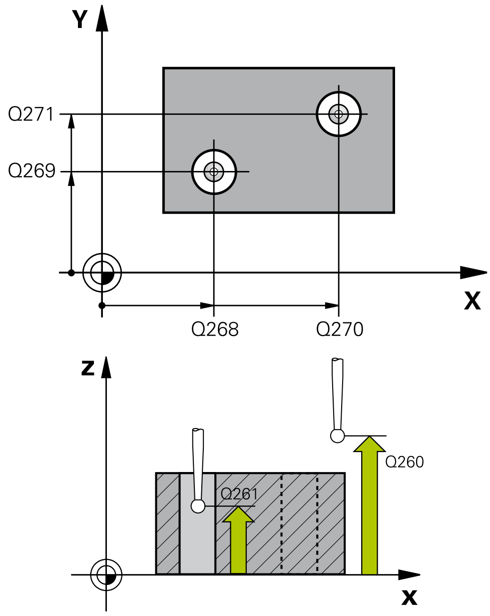
If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **1st hole: Center in 1st axis** Q268 (absolute): Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st hole: Center in 2nd axis** Q269 (absolute): Center of the first hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 1st axis** Q270 (absolute): Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 2nd axis** Q271 (absolute): Center of the second hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Preset value for rotation angle** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (**Q402=1**). In this case the misalignment is not saved as an angular value. Input range 0 to 99999



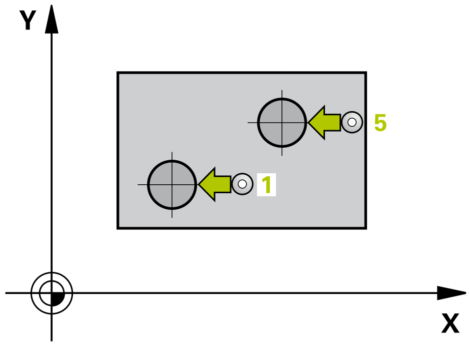
NC blocks

5 TCH PROBE 401 ROT OF 2 HOLES	
Q268=-37	;1ST CENTER IN 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+20	;CLEARANCE HEIGHT
Q307=0	;PRESET ROT. ANGLE
Q305=0	;NO. IN TABLE
Q402=0	;COMPENSATION
Q337=0	;ZERO RESET

- **Compensation** Q402: Define whether the TNC should set the measured misalignment as basic rotation or should align via rotating the rotary table:
  0: Set basic rotation
  1: Rotate the rotary table
  If you specify rotating the rotary table, the TNC does not save the measured misalignment, even if you have defined a table row in parameter **Q305**.
- **Set to zero after alignment** Q337: Define whether the TNC should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment:
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment. The TNC sets the display to 0 only if you have defined **Q402=1.**

14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402)

Cycle run



The Touch Probe Cycle 402 measures the centers of two studs. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the two stud centers. With the basic rotation function, the TNC compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 444) to touch point **1** of the first stud.
2 Then the probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves on a circular arc between the touch points, each of which is offset by 90°.
3 The touch probe returns to the clearance height and then positions the probe to starting point **5** of the second stud.
4 The probe moves to the entered **measuring height 2** and probes four points to find the center of the second stud.
5 Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

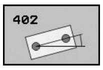
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC will reset an active basic rotation at the beginning of the cycle.

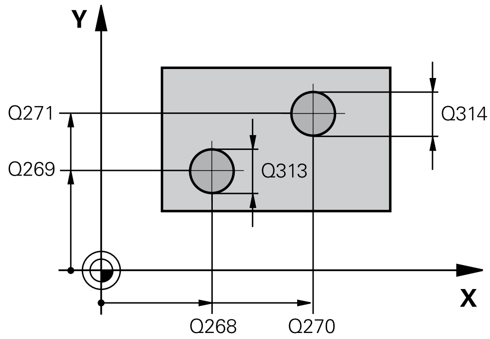
If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

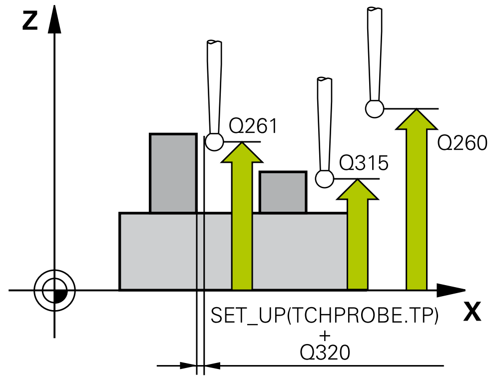
- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **1st stud: Center in 1st axis** Q268 (absolute): Center of the first stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st stud: Center in 2nd axis** Q269 (absolute): Center of the first stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Diameter of stud 1** Q313: Approximate diameter of the 1st stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Measuring height 1 in the probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 1 is to be measured. Input range -99999.9999 to 99999.9999
- **2nd stud: Center in 1st axis** Q270 (absolute): Center of the second stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd stud: Center in 2nd axis** Q271 (absolute): Center of the second stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Diameter of stud 2** Q314: Approximate diameter of the 2nd stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Measuring height of stud 2 in the probe axis** Q315 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 2 is to be measured. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points





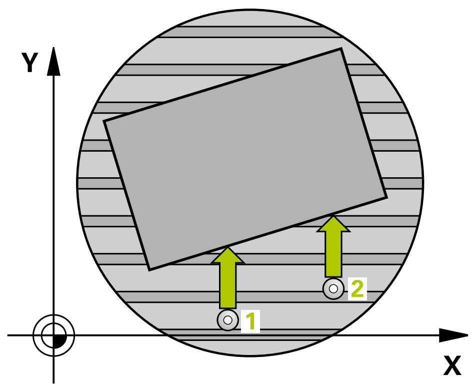
NC blocks

5 TCH PROBE 402 ROT OF 2 STUDS	
Q268=-37	;1ST CENTER IN 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q313=60	;DIAMETER OF STUD 1
Q261=-5	;MEASURING HEIGHT 1
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q314=60	;DIAMETER OF STUD 2
Q315=-5	;MEASURING HEIGHT 2
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q307=0	;PRESET ROT. ANGLE
Q305=0	;NO. IN TABLE
Q402=0	;COMPENSATION
Q337=0	;ZERO RESET

- **Preset value for rotation angle** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (**Q402=1**). In this case the misalignment is not saved as an angular value. Input range 0 to 99999
- **Compensation** Q402: Define whether the TNC should set the measured misalignment as basic rotation or should align via rotating the rotary table:
  0: Set basic rotation
  1: Rotate the rotary table
  If you specify rotating the rotary table, the TNC does not save the measured misalignment, even if you have defined a table row in parameter **Q305**.
- **Set to zero after alignment** Q337: Define whether the TNC should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment:
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment. The TNC sets the display to 0 only if you have defined **Q402=1.**

14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ ISO: G403)

Cycle run



Touch probe cycle 403 determines a workpiece misalignment by measuring two points, which must lie on a straight line. The TNC compensates the determined misalignment by rotating the A, B or C axis. The workpiece can be clamped in any position on the rotary table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the programmed touch point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves to the next starting position **2** and probes the second position.
4 The TNC returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally you can specify whether the TNC is to set the determined rotation angle to 0 in the preset table or in the datum table.

Please note while programming:

Danger of collision!

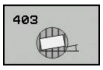
Ensure that the clearance height is sufficiently large so that no collisions can occur during the final positioning of the rotary axis.

If you enter 0 in parameter **Q312 Axis for compensating movement**, the cycle automatically determines the rotary axis to be aligned (recommended setting). Depending on the sequence of the probing points, an angle with the actual direction is determined. The measured angle goes from the first to the second probing point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the probing points. The calculated angle lies in the range from –90° to +90°. After alignment, check the position of the rotary axis.

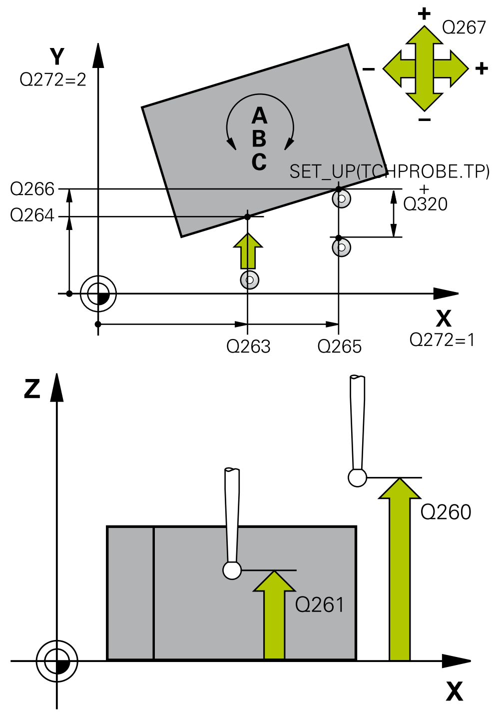
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC stores the measured angle in parameter **Q150**.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis (1...3: 1 = principal axis)** Q272: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative Traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999



NC blocks

5 TCH PROBE 403 ROT IN ROTARY AXIS	
Q263=+0	;1ST POINT 1ST AXIS
Q264=+0	;1ST POINT 2ND AXIS
Q265=+20	;2ND POINT 1ST AXIS
Q266=+30	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE

- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Axis for compensation movement** Q312: Assignment of the rotary axis in which the TNC is to compensate the measured misalignment:
  0: Automatic mode – the TNC uses the active kinematics to determine the rotary axis to be aligned. In automatic mode the first rotary table axis (as viewed from the workpiece) is used as compensation axis. Recommended setting.
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C
- **Set to zero after alignment** Q337: Define whether the TNC should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment:
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment
- **Number in table** Q305: Enter the number in the preset table/datum table in which the TNC is to set the rotary axis to zero. Only effective if Q337 is set to 1. Input range 0 to 99999
- **Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as datum shift active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).
- **Reference angle? (0=ref. axis)** Q380: Angle with which the TNC is to align the probed straight line. Only effective if rotary axis = automatic mode is selected, or rotary axis = C is selected (Q312 = 0 or 6). Input range -360.000 to 360.000

Q312=0	;COMPENSATION AXIS
Q337=0	;ZERO RESET
Q305=1	;NO. IN TABLE
Q303=+1	;MEAS. VALUE TRANSFER
Q380=+90	;REFERENCE ANGLE

14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)

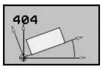
Cycle run

With Touch Probe Cycle 404, during program run you can automatically set any basic rotation or save it to the preset table. You can also use Cycle 404 if you want to reset an active basic rotation.

NC blocks

```
5 TCH PROBE 404 BASIC ROTATION
    Q307=+0    ;PRESET ROTATION
               ANG.
    Q305=-1    ;NO. IN TABLE
```

Cycle parameters



- **Preset value for rotation angle**: Angular value at which the basic rotation is to be set. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. Input range -1 to 99999. If you enter Q305=0 or Q305=1, the TNC additionally saves the determined basic rotation in the basic rotation menu (**PROBING ROT**) of the **Manual Operation** mode.
  -1 = Overwrite and activate the active preset
  0 = Copy the active preset to preset line 0, write the basic rotation to preset line 0 and activate preset 0
  >1 = Save the basic rotation to the specified preset. The preset is not activated.

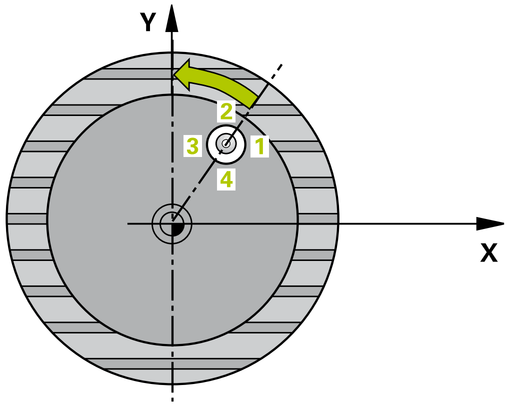
14.7 Compensating workpiece misalignment by rotating the C axis (Cycle 405, DIN/ISO: G405)

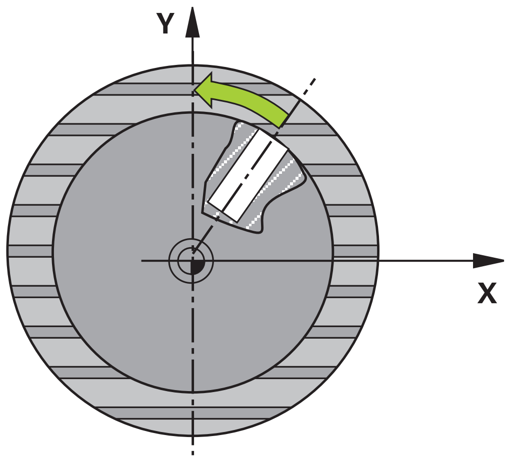
Cycle run

With Touch Probe Cycle 405, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center of a hole, or
- the angular offset between the nominal position and the actual position of a hole center.

The TNC compensates the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive. If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.





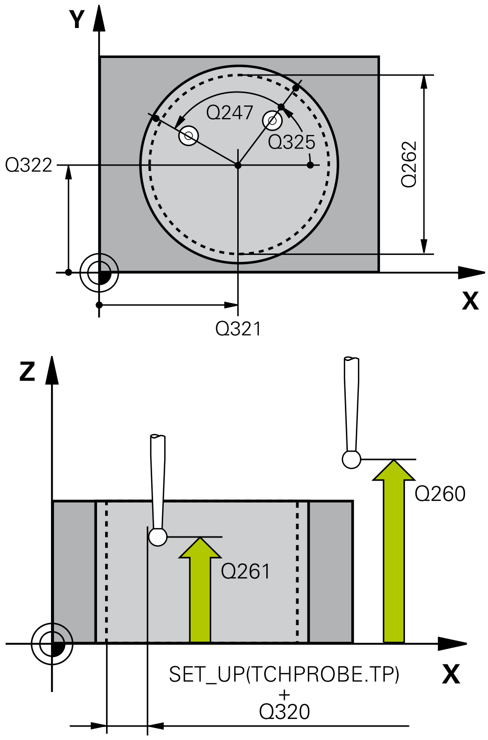
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points and positions the touch probe on the hole center measured.
5 Finally the TNC returns the touch probe to the clearance height and and aligns the workpiece by rotating the table. The TNC rotates the rotary table so that the hole center after compensation lies in the direction of the positive Y axis, or on the nominal position of the hole center—both with a vertical and horizontal touch probe axis. The measured angular misalignment is also available in parameter Q150.

Please note while programming:

> **!**
>
> **Danger of collision!**
>
> To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).
>
> If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.
>
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.
>
> The smaller the angle, the less accurately the TNC can calculate the circle center. Minimum input value: 5°.

Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute value): Center of the hole in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position (angle of the hole center). Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

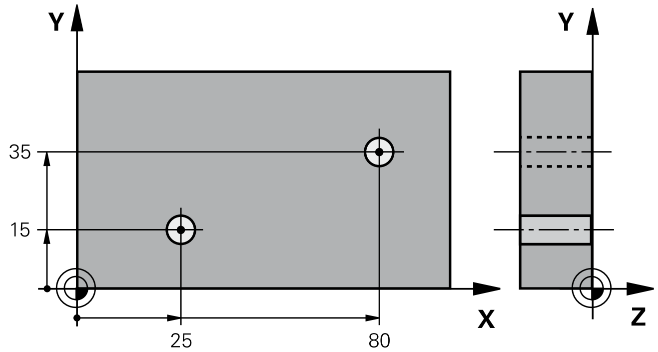


NC blocks

5 TCH PROBE 405 ROT IN C AXIS	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=10	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=90	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q337=0	;ZERO RESET

- **Set to zero after alignment** Q337: definition of whether the TNC should set the display of the C-axis to zero, or write the angular misalignment in column C of the datum table:
  0: Set the display of the C-axis to zero
  >0: Write the measured angular misalignment with correct algebraic signs in the datum table. Line number = value of Q337. If a C-axis shift is registered in the datum table, the TNC adds the measured angular misalignment.

14.8 Example: Determining a basic rotation from two holes



0 BEGIN PGM CYC401 MM	
1 TOOL CALL 69 Z	
2 TCH PROBE 401 ROT OF 2 HOLES	
Q268=+25 ;1ST CENTER 1ST AXIS	Center of the 1st hole: X coordinate
Q269=+15 ;1ST CENTER 2ND AXIS	Center of the 1st hole: Y coordinate
Q270=+80 ;2ND CENTER 1ST AXIS	Center of the 2nd hole: X coordinate
Q271=+35 ;2ND CENTER 2ND AXIS	Center of the 2nd hole: Y coordinate
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+20 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q307=+0 ;PRESET ROT. ANGLE	Angle of the reference line
Q402=1 ;COMPENSATION	Compensate misalignment by rotating the rotary table
Q337=1 ;ZERO RESET	Set the display to zero after the alignment
3 CALL PGM 35K47	Call part program
4 END PGM CYC401 MM	

15

Touch Probe Cycles: Automatic Datum Setting

15.1 Fundamentals

Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

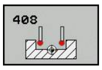
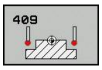
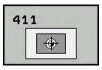
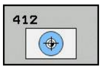
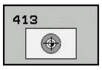
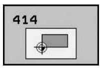
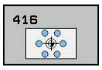
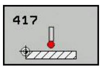
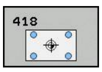
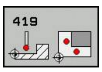
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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.

The TNC offers twelve cycles for automatically finding reference points and handling them as follows:

- Setting the determined values directly as display values
- Entering the determined values in the preset table
- Entering the determined values in a datum table

Cycle	Soft key	Page
408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as datum		474
409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as datum		478
410 DATUM INSIDE RECTANGLE Measuring the inside length and width of a rectangle, and defining the center as datum		481
411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as datum		485
412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum		488
413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum		493
414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as datum		497
415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as datum		502
416 DATUM CIRCLE CENTER (2nd soft-key level) Measuring any three holes on a bolt hole circle, and defining the bolt-hole center as datum		506
417 DATUM IN TS AXIS (2nd soft-key level) Measuring any position in the touch probe axis and defining it as datum		510
418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum		512
419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as datum		515

Characteristics common to all touch probe cycles for datum setting

You can also run the Touch Probe Cycles 408 to 419 during an active rotation (basic rotation or Cycle 10).

Datum point and touch probe axis

From the touch probe axis that you have defined in the measuring program the TNC determines the working plane for the datum.

Active touch probe axis	Datum setting in
Z	X and Y
Y	Z and X
X	Y and Z

Saving the calculated datum

In all cycles for datum setting you can use the input parameters Q303 and Q305 to define how the TNC is to save the calculated datum:

- **Q305 = 0, Q303 = any value**: The TNC sets the calculated datum in the display. The new datum is active immediately. At the same time, the TNC saves the datum set in the display by the cycle in line 0 of the preset table.
- **Q305 not equal to 0, Q303 = –1**

> This combination can only occur if you
> - read in programs containing Cycles 410 to 418 created on a TNC 4xx
> - read in programs containing Cycles 410 to 418 created with an older software version on an iTNC 530
> - did not specifically define the measured-value transfer with parameter Q303 when defining the cycle.
>
> In these cases the TNC outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter Q303.

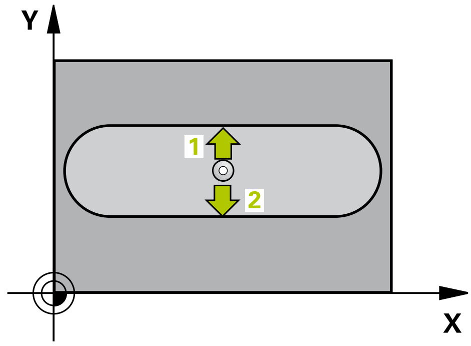
- **Q305 not equal to 0, Q303 = 0** The TNC writes the calculated reference point in the active datum table. The reference system is the active workpiece coordinate system. The value of parameter Q305 determines the datum number. **Activate the datum with Cycle 7 in the part program.**
- **Q305 not equal to 0, Q303 = 1** The TNC writes the calculated reference point in the preset table. The reference system is the machine coordinate system (REF coordinates). The value of parameter Q305 determines the preset number. **Activate the preset with Cycle 247 in the part program.**

Measurement results in Q parameters

The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. You can use these parameters in your program. Note the table of result parameters listed with every cycle description.

15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408)

Cycle run



Touch Probe Cycle 408 finds the center of a slot and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "") and saves the actual values in the Q parameters listed below.
5 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Please note while programming:

!

Danger of collision!

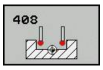
To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width.

If the slot width and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the slot. In this case the touch probe does not return to the clearance height between the two measuring points.

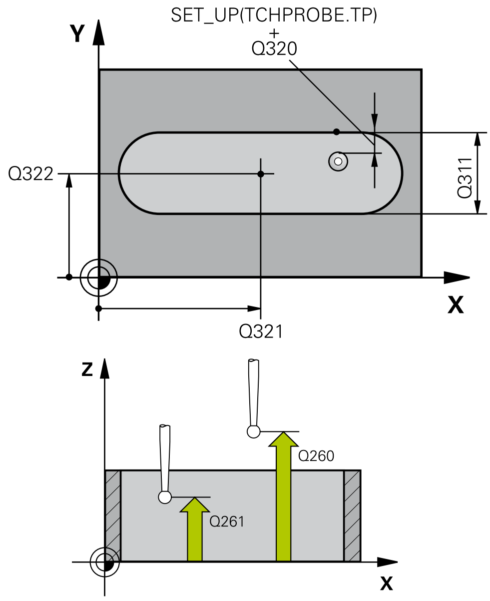
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the slot in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the slot in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Width of slot** Q311 (incremental): Width of the slot, regardless of its position in the working plane. Input range 0 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the slot center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the slot center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated slot center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as datum shift active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).



NC blocks

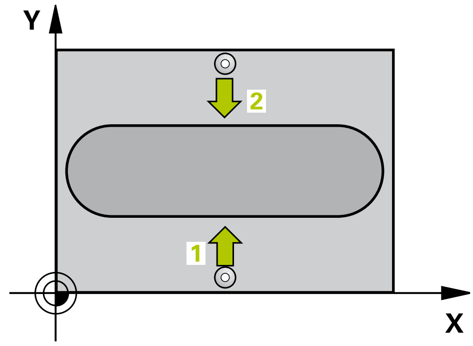
5 TCH PROBE 408 SLOT CENTER REF PT	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q311=25	;SLOT WIDTH
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=10	;NO. IN TABLE
Q405=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409)

Cycle run

Touch Probe Cycle 409 finds the center of a ridge and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves at clearance height to the next touch point **2** and probes the second touch point.
4 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the actual values in the Q parameters listed below.
5 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

Please note while programming:

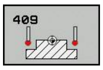
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

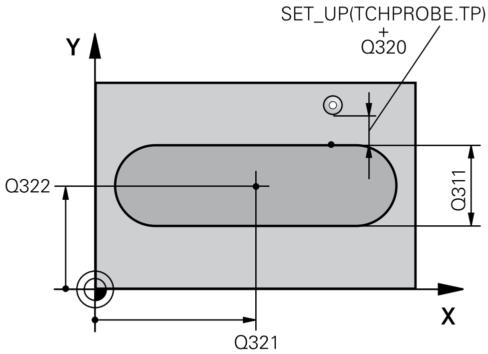
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

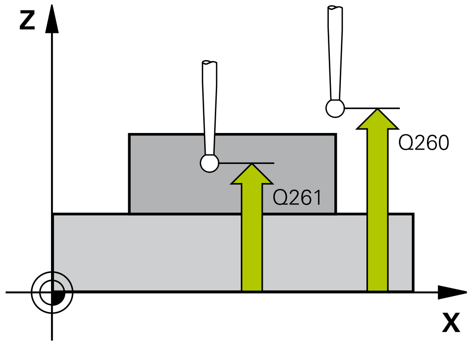
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the ridge in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the ridge in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Width of ridge** Q311 (incremental): Width of the ridge, regardless of its position in the working plane. Input range 0 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the ridge center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the ridge center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated ridge center. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as datum shift active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis





NC blocks

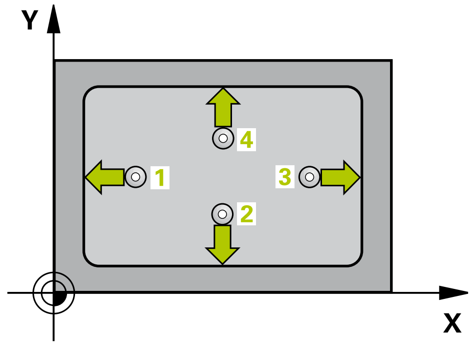
```
5 TCH PROBE 409 SLOT CENTER RIDGE
    Q321=+50   ;CENTER IN 1ST AXIS
    Q322=+50   ;CENTER IN 2ND AXIS
    Q311=25    ;SLOT WIDTH
    Q272=1     ;MEASURING AXIS
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q305=10    ;NO. IN TABLE
    Q405=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE
               TRANSFER
    Q381=1     ;PROBE IN TS AXIS
    Q382=+85   ;1ST CO. FOR TS AXIS
    Q383=+50   ;2ND CO. FOR TS AXIS
    Q384=+0    ;3RD CO. FOR TS AXIS
    Q333=+1    ;DATUM
```

- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

Cycle run

Touch Probe Cycle 410 finds the center of a rectangular pocket and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472).
6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis

Please note while programming:

!

Danger of collision!

To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides.

If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

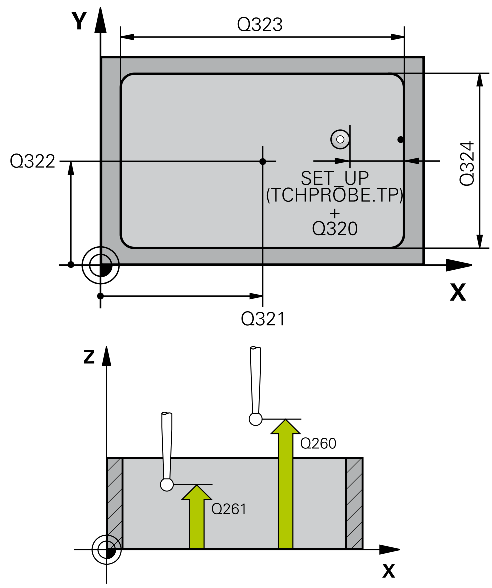
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Cycle parameters

410

- **Center in 1st axis** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q323 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0 input range -99999.9999 to 99999.9999



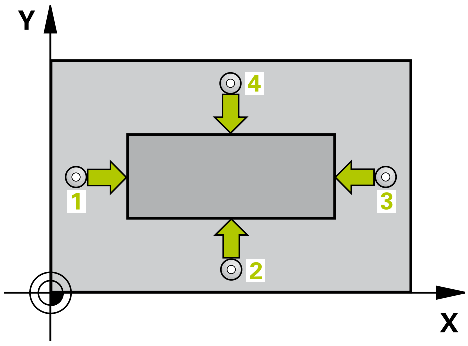
NC blocks

5 TCH PROBE 410 DATUM INSIDE RECTAN.	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;FIRST SIDE LENGTH
Q324=20	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=10	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM

- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0 input range -99999.9999 to 99999.9999

15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411)

Cycle run



Touch Probe Cycle 411 finds the center of a rectangular stud and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3. Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472).
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis

Please note while programming:

!

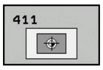
Danger of collision!

To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

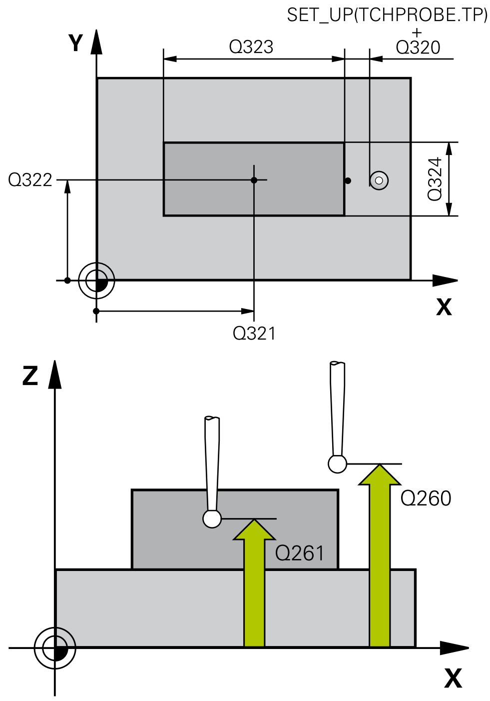
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q323 (incremental): Stud length, parallel to the reference axis of the working plane Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the stud center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999



NC blocks

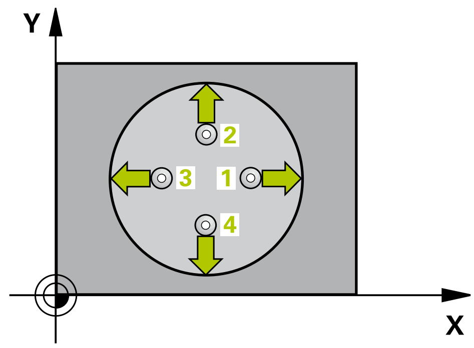
5 TCH PROBE 411 DATUM OUTS. RECTAN.	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;FIRST SIDE LENGTH
Q324=20	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=0	;NO. IN TABLE
Q331=+0	;DATUM

- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0 input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

Q332=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM

15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

Cycle run



Touch Probe Cycle 412 finds the center of a circular pocket (or of a hole) and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the actual values in the Q parameters listed below.
6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

Danger of collision!

To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

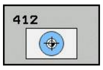
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

The smaller the angle increment Q247, the less accurately the TNC can calculate the datum. Minimum input value: 5°.

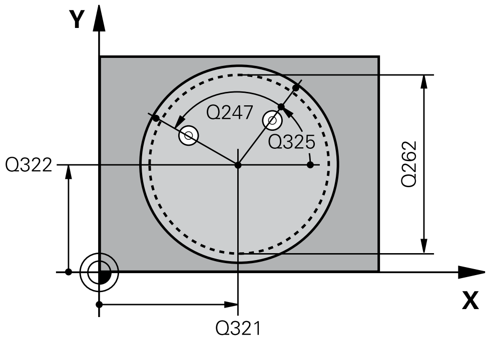
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

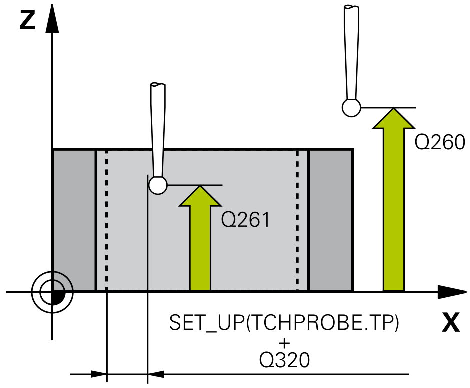
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the pocket in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999





NC blocks

5 TCH PROBE 412 DATUM INSIDE CIRCLE	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=12	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

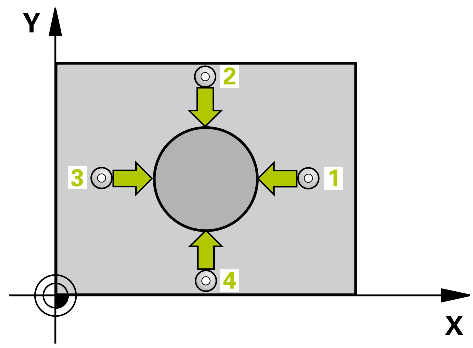
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)

Cycle run

Touch Probe Cycle 413 finds the center of a circular stud and defines it as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the actual values in the Q parameters listed below.
6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

!

Danger of collision!

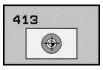
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

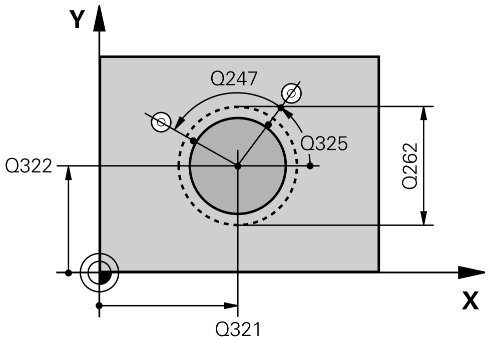
The smaller the angle increment Q247, the less accurately the TNC can calculate the datum. Minimum input value: 5°.

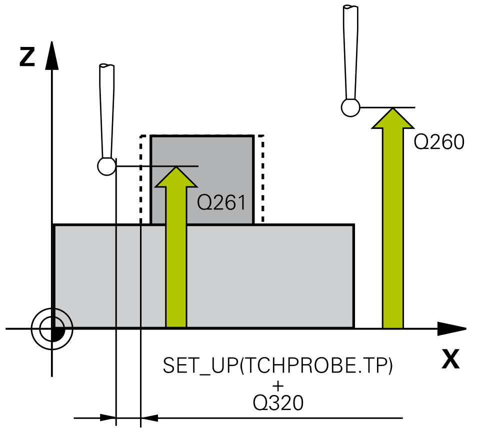
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points





NC blocks

5 TCH PROBE 413 DATUM OUTSIDE CIRCLE	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=15	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM

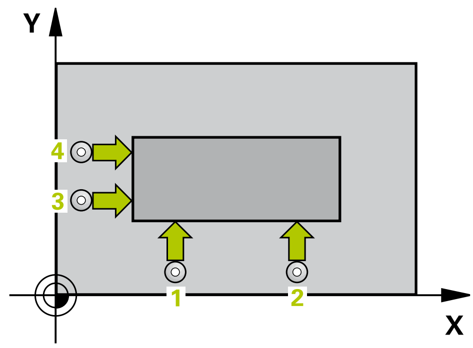
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the stud center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0 input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414)

Cycle run



Touch Probe Cycle 414 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1** (see figure at upper right). The TNC offsets the touch probe by the safety clearance in the direction opposite to the respective traverse direction.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The TNC derives the probing direction automatically from the programmed 3rd measuring point.
1 Then the touch probe moves to the next starting position **2** and from there probes the second position.
2 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
3 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the coordinates of the determined corner in the Q parameters listed below.
4 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

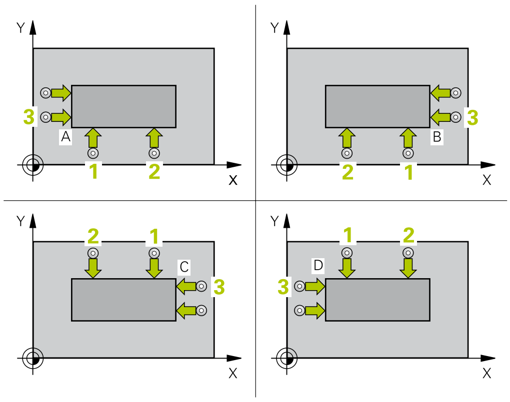
Danger of collision!

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

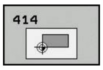
The TNC always measures the first line in the direction of the minor axis of the working plane.

By defining the positions of the measuring points **1** and **3** you also determine the corner at which the TNC sets the datum (see figure at right and table below).

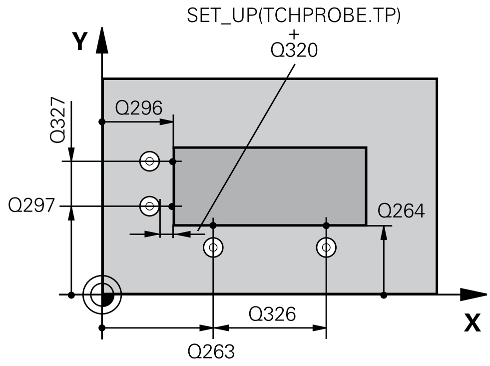


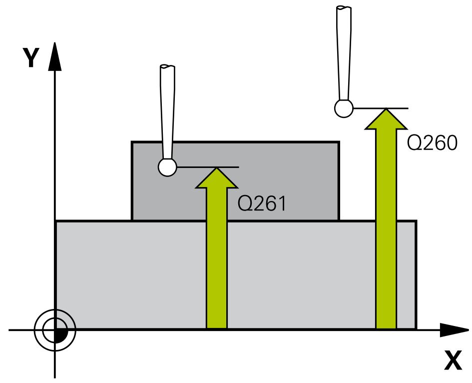
Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- **3rd meas. point 1st axis** Q296 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd meas. point 2nd axis** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points





NC blocks

5 TCH PROBE 414 DATUM INSIDE CORNER	
Q263=+37	;1ST POINT 1ST AXIS
Q264=+7	;1ST POINT 2ND AXIS
Q326=50	;SPACING IN 1ST AXIS
Q296=+95	;3RD POINT 1ST AXIS
Q297=+25	;3RD POINT 2ND AXIS
Q327=45	;SPACING IN 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q304=0	;BASIC ROTATION
Q305=7	;NO. IN TABLE
Q331=+0	;DATUM

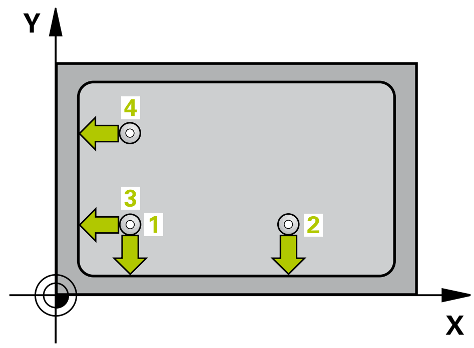
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation
- **Datum number in table** Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0 input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

Q332=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM

- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415)

Cycle run



Touch Probe Cycle 415 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1** (see figure at upper right) that you have defined in the cycle. The TNC offsets the touch probe by the safety clearance in the direction opposite to the respective traverse direction.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The probing direction is derived from the number by which you identify the corner.
1 Then the touch probe moves to the next starting position **2** and from there probes the second position.
2 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
3 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the coordinates of the determined corner in the Q parameters listed below.
4 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

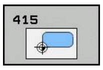
Danger of collision!

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

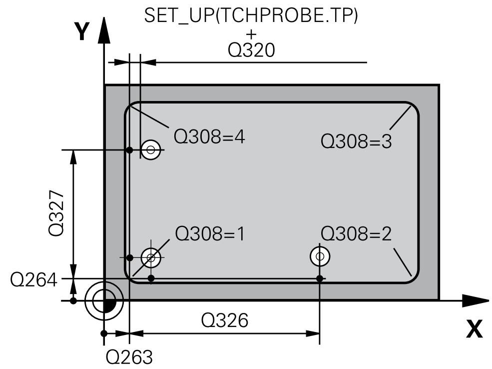
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

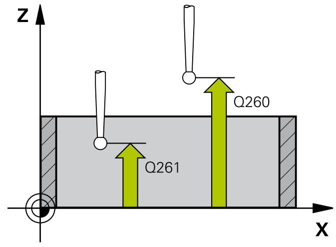
The TNC always measures the first line in the direction of the minor axis of the working plane.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- **Corner** Q308: Number identifying the corner which the TNC is to set as datum. Input range 1 to 4
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation
- **Datum number in table** Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999





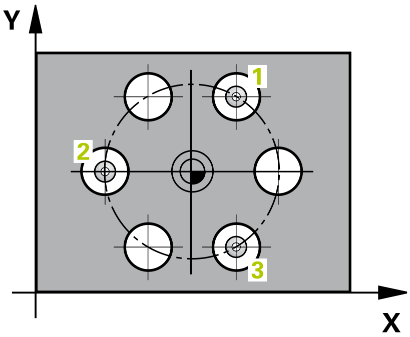
NC blocks

5 TCH PROBE 415 DATUM OUTSIDE CORNER	
Q263=+37	;1ST POINT 1ST AXIS
Q264=+7	;1ST POINT 2ND AXIS
Q326=50	;SPACING IN 1ST AXIS
Q296=+95	;3RD POINT 1ST AXIS
Q297=+25	;3RD POINT 2ND AXIS
Q327=45	;SPACING IN 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q304=0	;BASIC ROTATION
Q305=7	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM

- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0 input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)

Cycle run



Touch Probe Cycle 416 finds the center of a bolt hole circle and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the center of the first hole **1**.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6. The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the actual values in the Q parameters listed below.
8. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

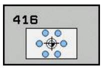
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Please note while programming:

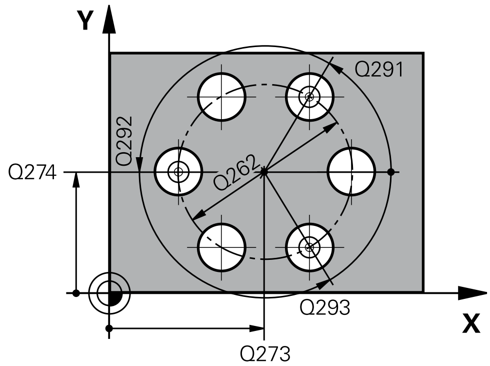
> **Danger of collision!**
> If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

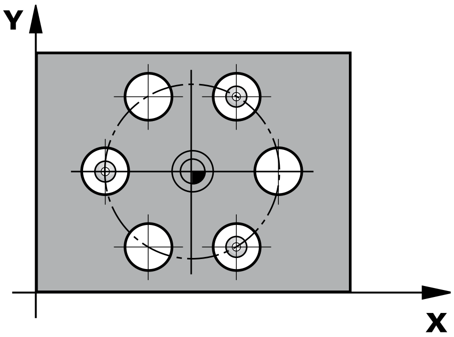
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input range 0 to 99999.9999
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the bolt-hole circle center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the bolt-hole circle center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the bolt-hole center. Default setting = 0 input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the bolt-hole center. Default setting = 0 input range -99999.9999 to 99999.9999





NC blocks

5 TCH PROBE 416 DATUM CIRCLE CENTER	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=90	;NOMINAL DIAMETER
Q291=+34	;ANGLE OF 1ST HOLE
Q292=+70	;ANGLE OF 2ND HOLE
Q293=+210	;ANGLE OF 3RD HOLE
Q261=-5	;MEASURING HEIGHT
Q260=+20	;CLEARANCE HEIGHT
Q305=12	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS

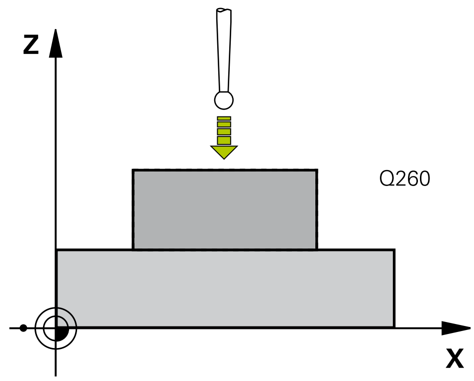
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the datum is probed in the touch probe axis. Input range 0 to 99999.9999

Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM
Q320=0	;SET-UP CLEARANCE

15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417)

Cycle run

Touch Probe Cycle 417 measures any coordinate in the touch probe axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.



1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the programmed touch point **1**. The TNC offsets the touch probe by the safety clearance in the positive direction of the touch probe axis.
2 Then the touch probe moves in its own axis to the coordinate entered as starting point **1** and measures the actual position with a simple probing movement.
3 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472) and saves the actual value in the Q parameters listed below.

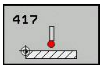
Parameter number	Meaning
Q160	Actual value of measured point

Please note while programming:

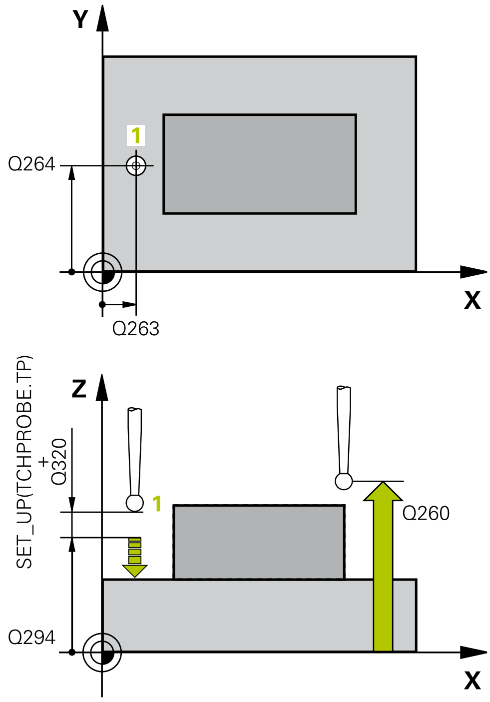
Danger of collision!
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
The TNC then sets the datum in this axis.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 3rd axis** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

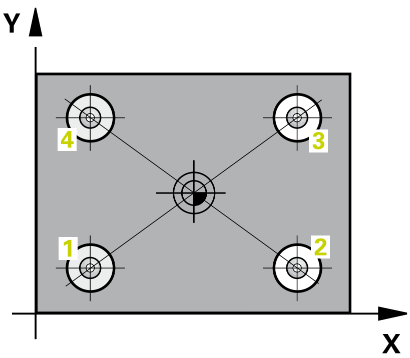


NC blocks

5 TCH PROBE 417 DATUM IN TS AXIS	
Q263=+25	;1ST POINT 1ST AXIS
Q264=+25	;1ST POINT 2ND AXIS
Q294=+25	;1ST POINT 3RD AXIS
Q320=0	;SET-UP CLEARANCE
Q260=+50	;CLEARANCE HEIGHT
Q305=0	;NO. IN TABLE
Q333=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER

15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)

Cycle run



Touch Probe Cycle 418 calculates the intersection of the lines connecting opposite holes and sets the datum at the intersection. If desired, the TNC can also enter the intersection into a datum table or preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the center of the first hole **1**.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The TNC repeats steps 3 and 4 for holes **3** and **4**.
6. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472). The TNC calculates the datum as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
7. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

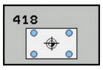
Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Please note while programming:

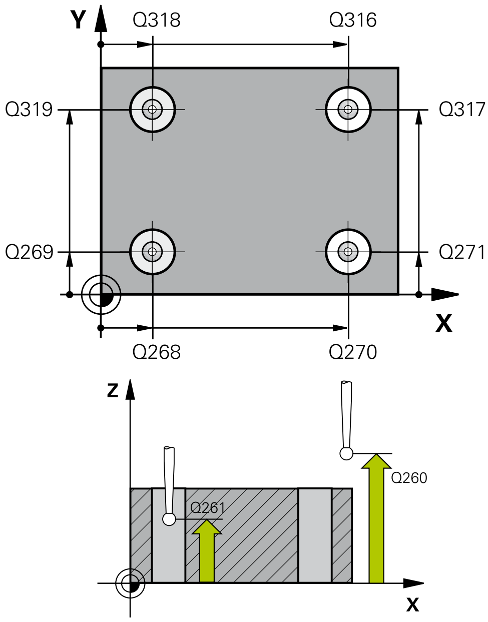
Danger of collision!
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **1st hole: Center in 1st axis** Q268 (absolute): Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st hole: Center in 2nd axis** Q269 (absolute): Center of the first hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 1st axis** Q270 (absolute): Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 2nd axis** Q271 (absolute): Center of the second hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd center in 1st axis** Q316 (absolute): center of the 3rd hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd center in 2nd axis** Q317 (absolute): center of the 3rd hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th center in 1st axis** Q318 (absolute): center of the 4th hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th center in 2nd axis** Q319 (absolute): center of the 4th hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the line intersection. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0 input range -99999.9999 to 99999.9999



NC blocks

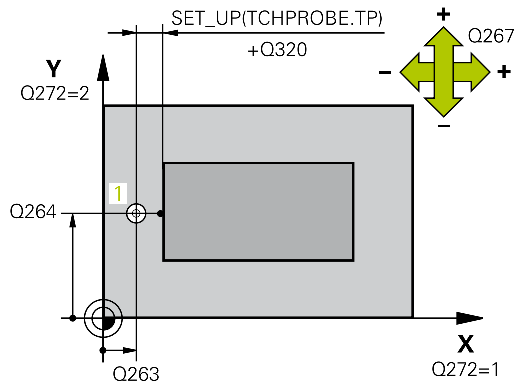
```
5 TCH PROBE 418 DATUM FROM 4
   HOLES
   Q268=+20  ;1ST CENTER 1ST AXIS
   Q269=+25  ;1ST CENTER 2ND AXIS
   Q270=+150;2ND CENTER 1ST AXIS
   Q271=+25  ;2ND CENTER 2ND AXIS
   Q316=+150;3RD CENTER 1ST AXIS
   Q317=+85  ;3RD CENTER 2ND AXIS
   Q318=+22  ;4TH CENTER 1ST AXIS
   Q319=+80  ;4TH CENTER 2ND AXIS
   Q261=-5   ;MEASURING HEIGHT
   Q260=+10  ;CLEARANCE HEIGHT
   Q305=12   ;NO. IN TABLE
   Q331=+0   ;DATUM
   Q332=+0   ;DATUM
   Q303=+1   ;MEAS. VALUE
              TRANSFER
   Q381=1    ;PROBE IN TS AXIS
   Q382=+85  ;1ST CO. FOR TS AXIS
   Q383=+50  ;2ND CO. FOR TS AXIS
   Q384=+0   ;3RD CO. FOR TS AXIS
   Q333=+0   ;DATUM
```

- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

Cycle run

Touch Probe Cycle 419 measures any coordinate in any axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.



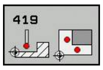
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the programmed touch point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite the programmed probing direction.
2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
3 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 472).

Please note while programming:

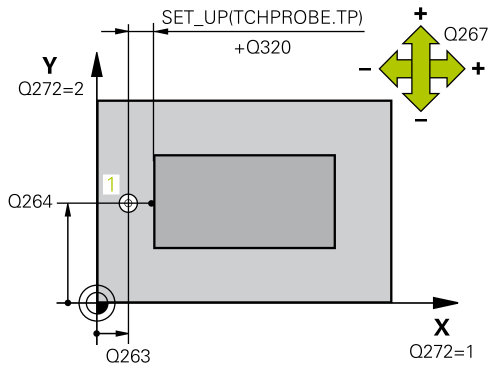
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

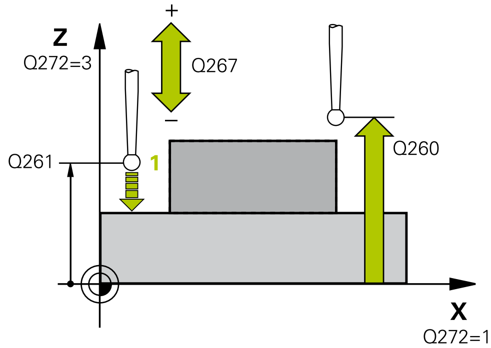
If you use Cycle 419 several times in succession to save the datum in more than one axis in the preset table, you must activate the preset number last written to by Cycle 419 after every execution of Cycle 419 (this is not required if you overwrite the active preset).

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Measuring axis (1...3: 1 = principal axis)** Q272: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis





Axis assignment

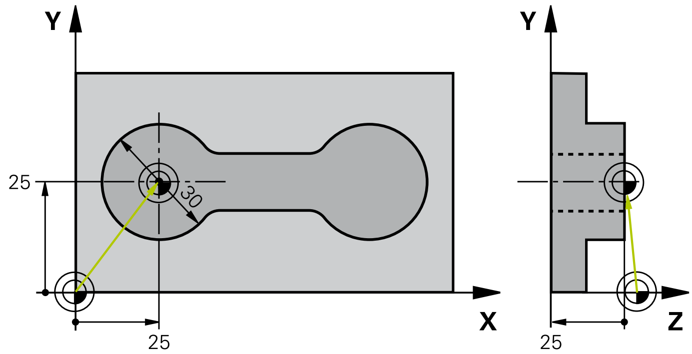
Active touch probe axis: Q272= 3	Corresponding reference axis: Q272= 1	Corresponding minor axis: Q272= 2
Z	X	Y
Y	Z	X
X	Y	Z

NC blocks

```
5 TCH PROBE 419 DATUM IN ONE AXIS
    Q263=+25   ;1ST POINT 1ST AXIS
    Q264=+25   ;1ST POINT 2ND AXIS
    Q261=+25   ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+50   ;CLEARANCE HEIGHT
    Q272=+1    ;MEASURING AXIS
    Q267=+1    ;TRAVERSE DIRECTION
    Q305=0     ;NO. IN TABLE
    Q333=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE
               TRANSFER
```

- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative Traverse direction
  +1: Positive traverse direction
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 472)
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

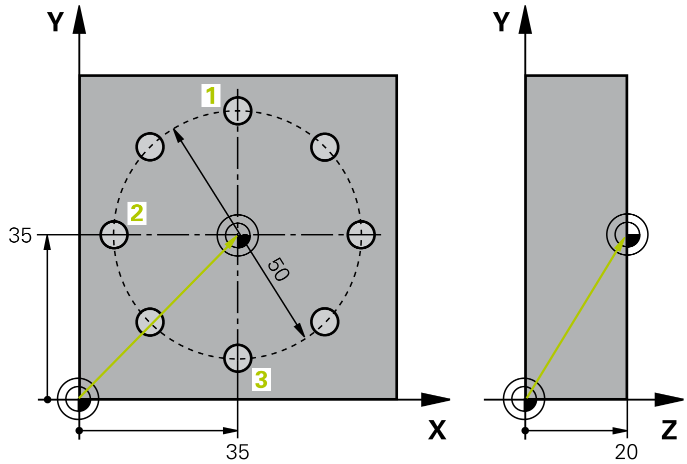
15.14 Example: Datum setting in center of a circular segment and on top surface of workpiece



0 BEGIN PGM CYC413 MM	
1 TOOL CALL 69 Z	Call tool 0 to define the touch probe axis
2 TCH PROBE 413 DATUM OUTSIDE CIRCLE	
Q321=+25 ;CENTER 1ST AXIS	Center of circle: X coordinate
Q322=+25 ;CENTER 2ND AXIS	Center of circle: Y coordinate
Q262=30 ;NOMINAL DIAMETER	Circle diameter
Q325=+90 ;STARTING ANGLE	Polar coordinate angle for 1st touch point
Q247=+45 ;STEPPING ANGLE	Stepping angle for calculating the starting points 2 to 4
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q320=2 ;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q301=0 ;MOVE TO CLEARANCE	Do not move to clearance height between measuring points
Q305=0 ;NO. IN TABLE	Set display
Q331=+0 ;DATUM	Set the display in X to 0
Q332=+10 ;DATUM	Set the display in Y to 10
Q303=+0 ;MEAS. VALUE TRANSFER	Without function, since display is to be set
Q381=1 ;PROBE IN TS AXIS	Also set datum in the touch probe axis
Q382=+25 ;1ST CO. FOR TS AXIS	X coordinate of touch point
Q383=+25 ;2ND CO. FOR TS AXIS	Y coordinate of touch point
Q384=+25 ;3RD CO. FOR TS AXIS	Z coordinate of touch point
Q333=+0 ;DATUM	Set the display in Z to 0
Q423=4 ;NO. OF PROBE POINTS	Measure circle with 4 probes
Q365=0 ;TYPE OF TRAVERSE	Move on circular path between measuring points
3 CALL PGM 35K47	Call part program
4 END PGM CYC413 MM	

15.15 Example: Datum setting on top surface of workpiece and in center of a bolt hole circle

The measured bolt hole center shall be written in the preset table so that it may be used at a later time.



0 BEGIN PGM CYC416 MM	
1 TOOL CALL 69 Z	Call tool 0 to define the touch probe axis
2 TCH PROBE 417 DATUM IN TS AXIS	Cycle definition for datum setting in the touch probe axis
Q263=+7.5 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=0 ;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+50 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NO. IN TABLE	Write Z coordinate in line 1
Q333=+0 ;DATUM	Set touch-probe axis to 0
Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)
3 TCH PROBE 416 DATUM CIRCLE CENTER	
Q273=+35 ;CENTER IN 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35 ;CENTER IN 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50 ;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90 ;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center 1
Q292=+180 ;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center 2
Q293=+270 ;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center 3
Q261=+15 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NO. IN TABLE	Enter center of bolt hole circle (X and Y) in line 1
Q331=+0 ;DATUM	
Q332=+0 ;DATUM	

Q303=+1	;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)
Q381=0	;PROBE IN TS AXIS	Do not set a datum in the touch probe axis
Q382=+0	;1ST CO. FOR TS AXIS	No function
Q383=+0	;2ND CO. FOR TS AXIS	No function
Q384=+0	;3RD CO. FOR TS AXIS	No function
Q333=+0	;DATUM	No function
Q320=0	;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
4 CYCL DEF 247 DATUM SETTING		Activate new preset with Cycle 247
Q339=1	;DATUM NUMBER	
6 CALL PGM 35KLZ		Call part program
7 END PGM CYC416 MM		

16

Touch Probe Cycles: Automatic Workpiece Inspection

16.1 Fundamentals

Overview

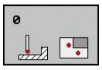
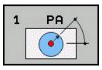
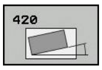
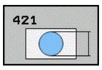
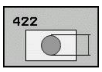
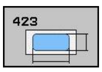
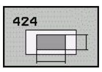
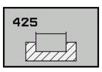
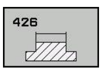
When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

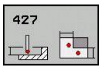
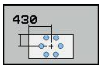
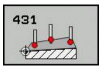
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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.

The TNC offers twelve cycles for measuring workpieces automatically.

Cycle	Soft key	Page
0 REFERENCE PLANE Measuring a coordinate in a selectable axis		528
1 POLAR DATUM PLANE Measuring a point in a probing direction		529
420 MEASURE ANGLE Measuring an angle in the working plane		530
421 MEASURE HOLE Measuring the position and diameter of a hole		533
422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud		536
423 MEASURE RECTANGLE INSIDE Measuring the position, length and width of a rectangular pocket		539
424 MEASURE RECTANGLE OUTSIDE Measuring the position, length and width of a rectangular stud		542
425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width		545
426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge		548

Cycle	Soft key	Page
427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis		551
430 MEASURE BOLT HOLE CIRCLE (2nd soft-key row) Measuring position and diameter of a bolt hole circle		554
431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane		557

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles 0 and 1), you can have the TNC record the measurement results. In the respective probing cycle you can define if the TNC is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file, the TNC, by default, saves the data as an ASCII file in the directory TNC:\..

Use the HEIDENHAIN data transfer software TNCRemo if you wish to output the measuring log over the data interface.

Example: Measuring log for touch probe cycle 421:

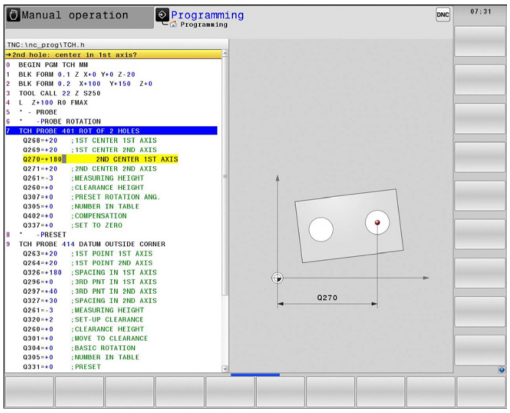
Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H

Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000
Actual values:	
Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259
Deviations:	
Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259
Further measuring results: Measuring height:	-5.0000

End of measuring log

Measurement results in Q parameters



The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. Deviations from the nominal value are saved in the parameters Q161 to Q166. Note the table of result parameters listed with every cycle description.

During cycle definition the TNC also shows the result parameters for the respective cycle in a help graphic (see figure at upper right). The highlighted result parameter belongs to that input parameter.

Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters Q180 to Q182.

Class of results	Parameter value
Measurement results are within tolerance	Q180 = 1
Rework is required	Q181 = 1
Scrap	Q182 = 1

The TNC sets the rework or scrap marker as soon as one of the measuring values falls outside of tolerance. To determine which of the measuring results lies outside of tolerance, check the measuring log, or compare the respective measuring results (Q150 to Q160) with their limit values.

In Cycle 427 the TNC assumes that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

> The TNC also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

For most of the cycles for workpiece inspection you can have the TNC perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the 0 (the default value) in the monitoring parameters.

Tool monitoring

For some cycles for workpiece inspection you can have the TNC perform tool monitoring. The TNC then monitors whether

- The tool radius should be compensated because of the deviations from the nominal value (values in Q16x).
- The deviations from the nominal value (values in Q16x) are greater than the tool breakage tolerance.

Tool compensation

> This function works only:
> - If the tool table is active.
> - If tool monitoring is switched on in the cycle (enter a tool name or **Q330** unequal to 0). Select the tool name input by soft key. The TNC no longer displays the right single quotation mark.
>
> If you perform several compensation measurements, the TNC adds the respective measured deviation to the value stored in the tool table.

The TNC always compensates the tool radius in the DR column of the tool table, even if the measured deviation lies within the given tolerance. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: must be reworked).

For Cycle 427:

- If an axis of the active working plane is defined as measuring axis (Q272 = 1 or 2), the TNC compensates the tool radius as described above. From the defined traversing direction (Q267) the TNC determines the direction of compensation.
- If the touch probe axis is defined as measuring axis (Q272 = 3), the TNC compensates the tool length.

Tool breakage monitoring

> This function works only:
> - If the tool table is active.
> - If tool monitoring is switched on in the cycle (enter Q330 not equal to 0).
> - If the breakage tolerance RBREAK for the tool number entered in the table is greater than 0 (see also the User's Manual, section 5.2 "Tool Data").

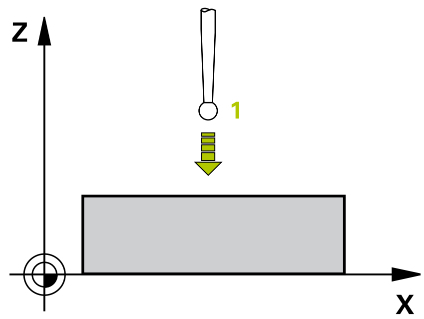
The TNC will output an error message and stop program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time the tool will be deactivated in the tool table (column TL = L).

Reference system for measurement results

The TNC transfers all the measurement results to the result parameters and the log file in the active coordinate system, or as the case may be, the shifted and/or rotated/tilted coordinate system.

16.2 DATUM PLANE (Cycle 0, DIN/ISO: G55)

Cycle run

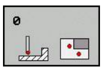


1 The touch probe moves at rapid traverse (value from **FMAX** column) to the starting position **1** programmed in the cycle.
2 Then the touch probe runs the probing process at the probing feed rate (column **F**). The probing direction is defined in the cycle.
3 After the TNC has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in the parameters Q115 to Q119. For the values in these parameters the TNC does not account for the stylus length and radius.

Please note while programming:

> **!** **Danger of collision!**
> Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

Cycle parameters



- **Parameter number for result:** Enter the number of the Q parameter to which you want to assign the coordinate. Input range 0 to 1999
- **Probing axis/Probing direction**: Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Confirm your entry with the **ENT** key. Input range: All NC axes
- **Nominal position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- To conclude the input, press the **ENT** key.

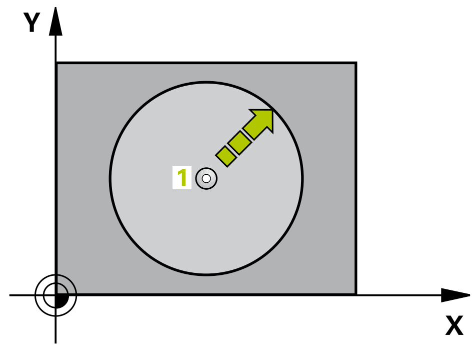
NC blocks

```
67 TCH PROBE 0.0 REF. PLANE Q5 X-
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

16.3 POLAR DATUM PLANE (Cycle 1)

Cycle run

Touch Probe Cycle 1 measures any position on the workpiece in any direction.



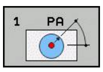
1 The touch probe moves at rapid traverse (value from **FMAX** column) to the starting position **1** programmed in the cycle.
2 Then the touch probe runs the probing process at the probing feed rate (column **F**). During probing the TNC moves simultaneously in two axes (depending on the probing angle). The probing direction is defined by the polar angle entered in the cycle.
3 After the TNC has saved the position, the probe returns to the starting point. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in parameters Q115 to Q119.

Please note while programming:

Danger of collision!
Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

The probing axis defined in the cycle specifies the probing plane:
Probing axis X: X/Y plane
Probing axis Y: Y/Z plane
Probing axis Z: Z/X plane

Cycle parameters



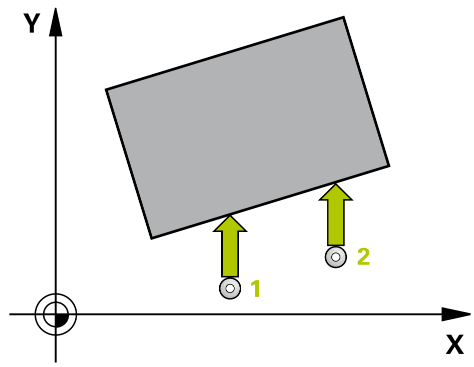
- **Probing axis:** Enter the probing axis with the axis selection keys or ASCII keyboard. Confirm your entry with the **ENT** key. Input range: **X**, **Y** or **Z**
- **Probing angle:** Angle, measured from the probing axis, at which the touch probe is to move. Input range -180.0000 to 180.0000
- **Nominal position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- To conclude the input, press the **ENT** key.

NC blocks

```
67 TCH PROBE 1.0 POLAR REFERENCE
   PLANE
68 TCH PROBE 1.1 X ANGLE: +30
69 TCH PROBE 1.2 X+5 Y+0 Z-5
```

16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

Cycle run



Touch Probe Cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the programmed touch point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves to the next starting position **2** and from there probes the second position.
4 The TNC returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

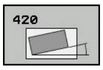
Parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the machining plane.

Please note while programming:

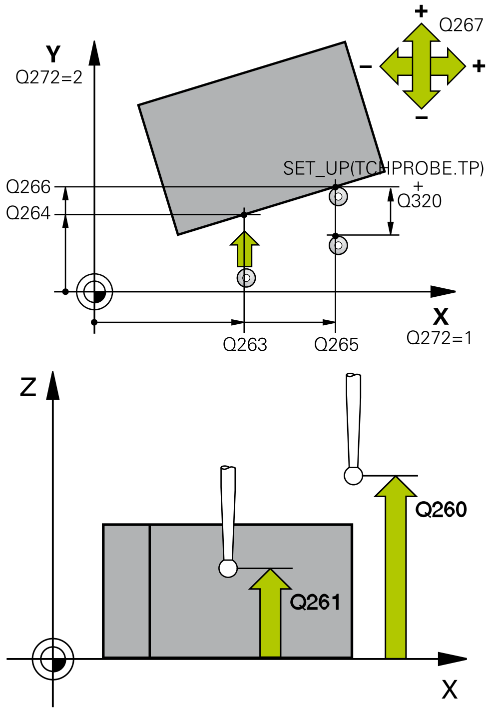
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If touch probe axis = measuring axis, set **Q263** equal to **Q265** if the angle about the A axis is to be measured; set **Q263** not equal to **Q265** if the angle is to be measured about the B axis.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points



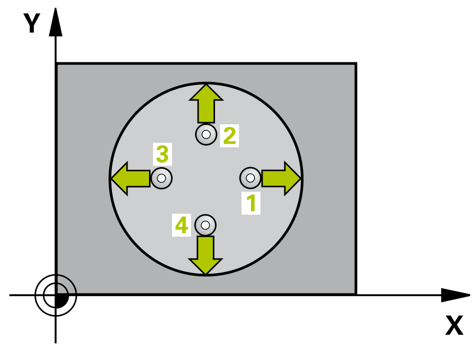
NC blocks

5 TCH PROBE 420 MEASURE ANGLE	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+10	;1ST POINT 2ND AXIS
Q265=+15	;2ND POINT 1ST AXIS
Q266=+95	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q281=1	;MEASURING LOG

- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR420.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.

16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)

Cycle run



Touch Probe Cycle 421 measures the center and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

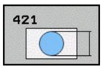
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

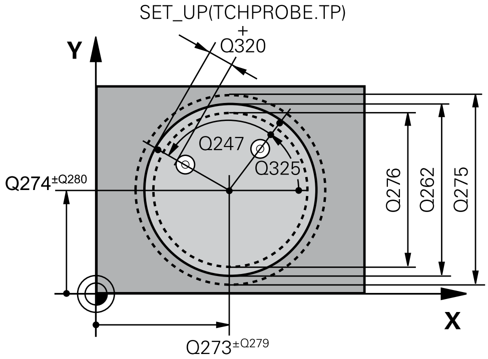
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

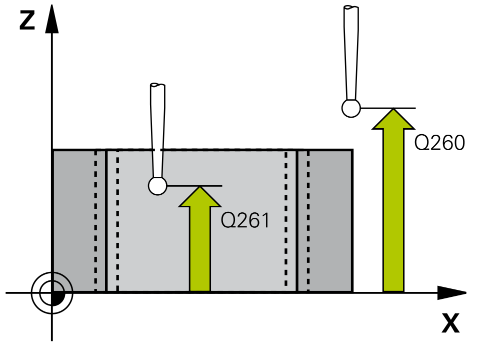
The smaller the angle, the less accurately the TNC can calculate the hole dimensions. Minimum input value: 5°

Cycle parameters



- **Center in 1st axis** Q273 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute value): Center of the hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the diameter of the hole. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Maximum limit of size for hole** Q275: Maximum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- **Minimum limit of size for hole** Q276: Minimum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999





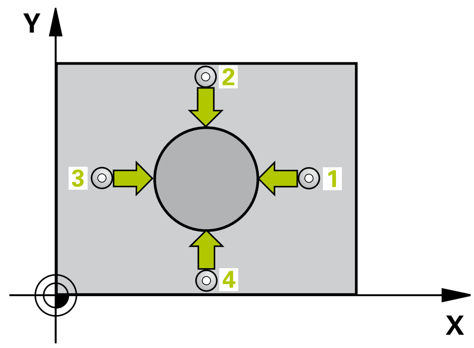
NC blocks

5 TCH PROBE 421 MEASURE HOLE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q275=75.12	;MAXIMUM DIMENSION
Q276=74.95	;MINIMUM DIMENSION
Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR421.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

Cycle run



Touch Probe Cycle 422 measures the center and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

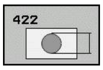
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

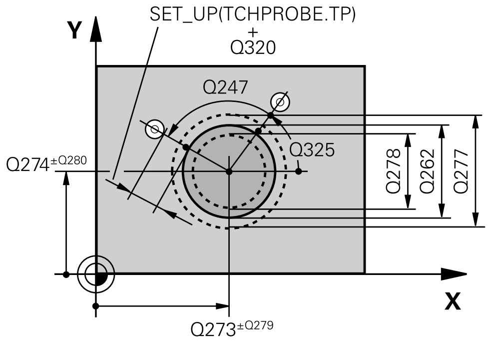
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

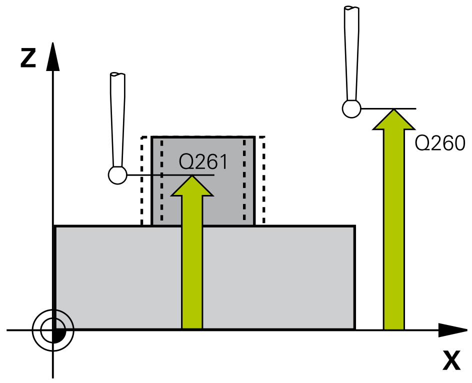
The smaller the angle, the less accurately the TNC can calculate the dimensions of the stud. Minimum input value: 5°

Cycle parameters



- **Center in 1st axis** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the diameter of the stud. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Maximum limit of size for stud** Q277: Maximum permissible diameter for the stud. Input range 0 to 99999.9999
- **Minimum limit of size for the stud** Q278: Minimum permissible diameter for the stud. Input range 0 to 99999.9999





NC blocks

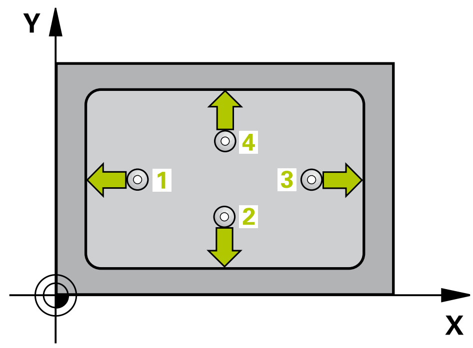
5 TCH PROBE 422 MEAS. CIRCLE OUTSIDE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+90	;STARTING ANGLE
Q247=+30	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q275=35.15	;MAXIMUM DIMENSION
Q276=34.9	;MINIMUM DIMENSION
Q279=0.05	;TOLERANCE 1ST CENTER

- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR422.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

Q280=0.05	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

16.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423)

Cycle run



Touch Probe Cycle 423 finds the center, length and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

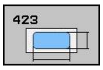
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in reference axis
Q165	Deviation of side length in minor axis

Please note while programming:

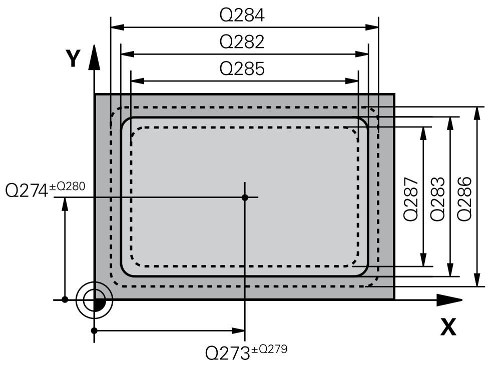
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

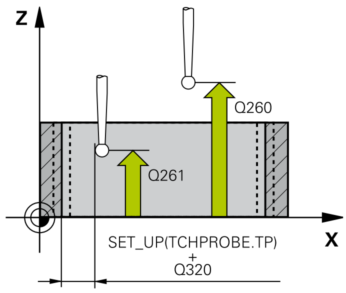
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

Cycle parameters



- **Center in 1st axis** Q273 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q282: Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q283: Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Max. size limit 1st side length** Q284: Maximum permissible length of the pocket. Input range 0 to 99999.9999
- **Min. size limit 1st side length** Q285: Minimum permissible length of the pocket. Input range 0 to 99999.9999
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the pocket. Input range 0 to 99999.9999
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the pocket. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999





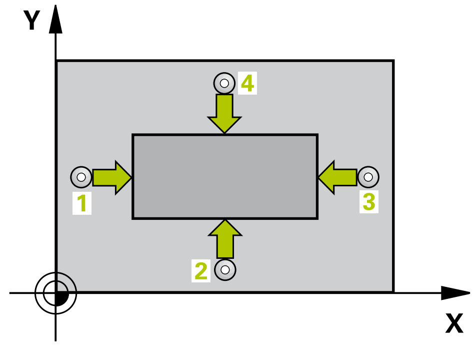
NC blocks

5 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q282=80	;FIRST SIDE LENGTH
Q283=60	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q284=0	;MAX. LIMIT 1ST SIDE
Q285=0	;MIN. LIMIT 1ST SIDE
Q286=0	;MAX. LIMIT 2ND SIDE
Q287=0	;MIN. LIMIT 2ND SIDE
Q279=0	;TOLERANCE 1ST CENTER
Q280=0	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL

- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR423.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T

16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)

Cycle run



Touch Probe Cycle 424 finds the center, length and width of a rectangular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

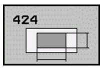
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in reference axis
Q165	Deviation of side length in minor axis

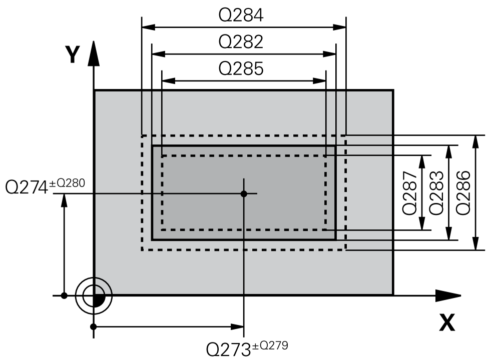
Please note while programming:

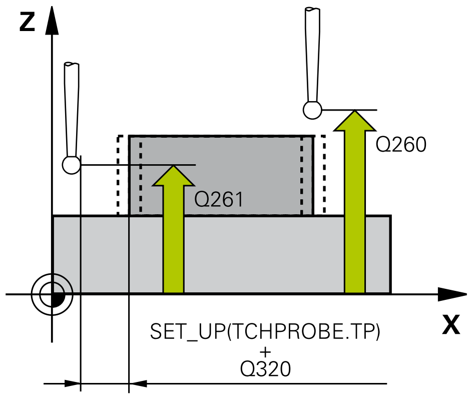
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Center in 1st axis** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q282: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q283: Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Max. size limit 1st side length** Q284: Maximum permissible length of the stud. Input range 0 to 99999.9999
- **Min. size limit 1st side length** Q285: Minimum permissible length of the stud. Input range 0 to 99999.9999
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the stud. Input range 0 to 99999.9999
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the stud. Input range 0 to 99999.9999





NC blocks

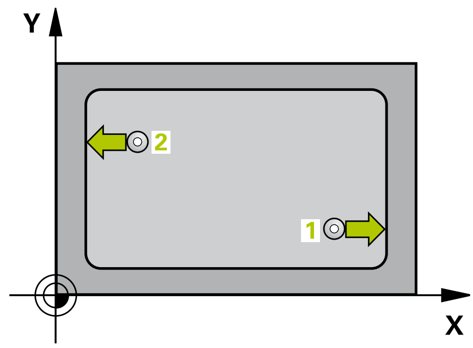
5 TCH PROBE 424 MEAS. RECTAN. OUTS.	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q282=75	;FIRST SIDE LENGTH
Q283=35	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q284=75.1	;MAX. LIMIT 1ST SIDE
Q285=74.9	;MIN. LIMIT 1ST SIDE
Q286=35	;MAX. LIMIT 2ND SIDE
Q287=34.95	;MIN. LIMIT 2ND SIDE

- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR424.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T

Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL

16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)

Cycle run



Touch Probe Cycle 425 measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in a system parameter.

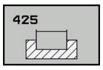
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). 1. The first probing is always in the positive direction of the programmed axis.
3 If you enter an offset for the second measurement, the TNC then moves the touch probe (if required, at clearance height) to the next starting point **2** and probes the second touch point. If the nominal length is large, the TNC moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the TNC measures the width in the exact opposite direction.
4 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation value in the following Q parameters:

Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

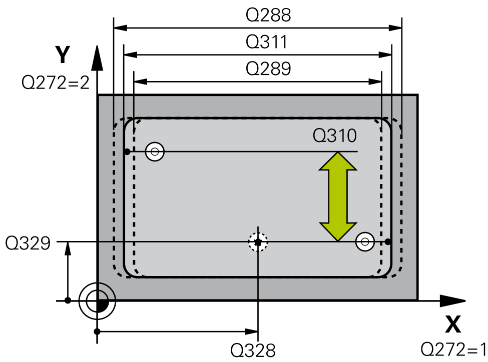
Please note while programming:

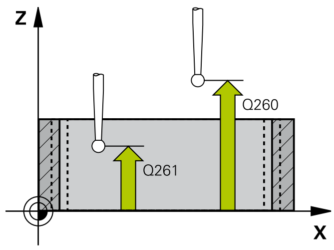
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Starting point in 1st axis** Q328 (absolute): Starting point for probing in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 2nd axis** Q329 (absolute): Starting point for probing in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Offset for 2nd measurement** Q310 (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the TNC does not offset the touch probe. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Nominal length** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999
- **Maximum dimension** Q288: Maximum permissible length. Input range 0 to 99999.9999
- **Minimum dimension** Q289: Minimum permissible length. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR425.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message





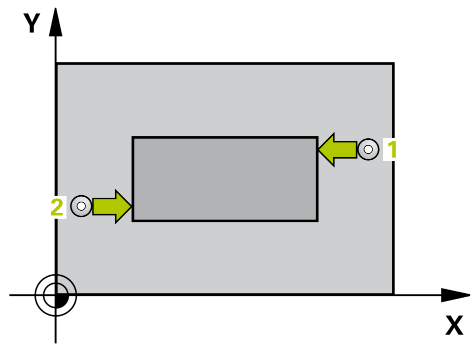
NC blocks

5 TCH PROBE 425 MEASURE INSIDE WIDTH	
Q328=+75	;STARTNG PNT 1ST AXIS
Q329=-12.5	;STARTNG PNT 2ND AXIS
Q310=+0	;OFFS. 2ND MEASUREMNT
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+10	;CLEARANCE HEIGHT
Q311=25	;NOMINAL LENGTH
Q288=25.05	;MAXIMUM DIMENSION
Q289=25	;MINIMUM DIMENSION
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL
Q320=0	;SET-UP CLEARANCE
Q301=0	;MOVE TO CLEARANCE

- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the datum is probed in the touch probe axis. Input range 0 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)

Cycle run



Touch Probe Cycle 426 measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

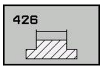
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC calculates the touch points from the data in the cycle and the safety clearance from the **SET_UP** column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**). 1. The first probing is always in the negative direction of the programmed axis.
3 Then the touch probe moves at clearance height to the next starting position and probes the second touch point.
4 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation value in the following Q parameters:

Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

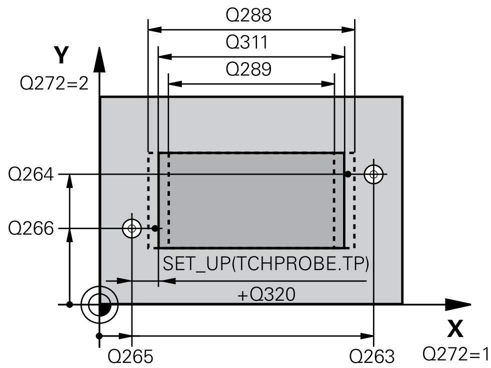
Please note while programming:

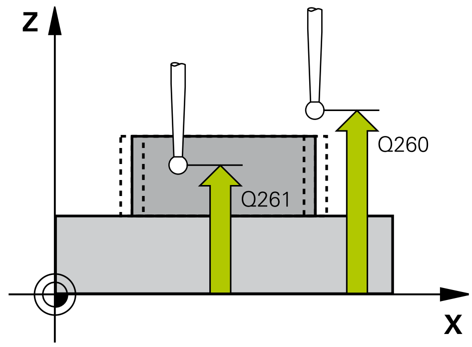
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Nominal length** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999
- **Maximum dimension** Q288: Maximum permissible length. Input range 0 to 99999.9999
- **Minimum dimension** Q289: Minimum permissible length. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR426.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.





NC blocks

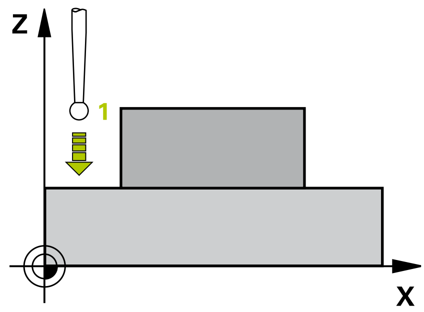
5 TCH PROBE 426 MEASURE RIDGE WIDTH	
Q263=+50	;1ST POINT 1ST AXIS
Q264=+25	;1ST POINT 2ND AXIS
Q265=+50	;2ND POINT 1ST AXIS
Q266=+85	;2ND POINT 2ND AXIS
Q272=2	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q311=45	;NOMINAL LENGTH
Q288=45	;MAXIMUM DIMENSION
Q289=44.95	;MINIMUM DIMENSION
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL

- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T

16.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)

Cycle run

Touch Probe Cycle 427 finds a coordinate in a selectable axis and saves the value in a system parameter. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



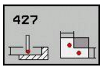
1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to touch point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2. Then the TNC positions the touch probe to the entered touch point **1** in the working plane and measures the actual value in the selected axis.
3. Finally the TNC returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter.

Parameter number	Meaning
Q160	Measured coordinate

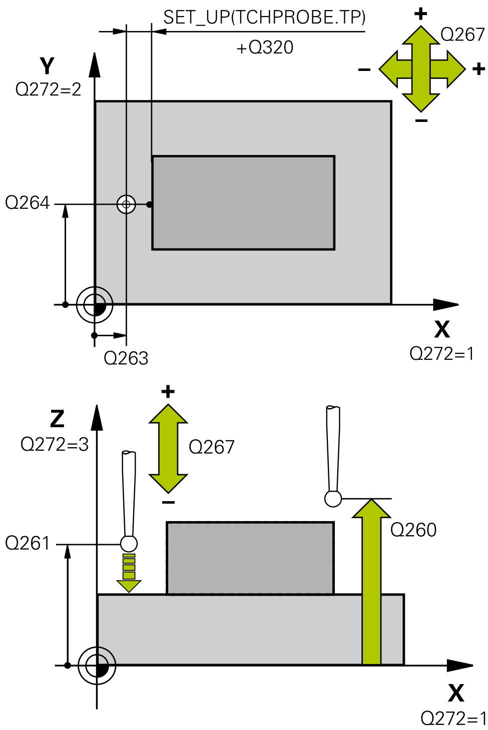
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Measuring axis (1 to 3: 1 = Reference axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR427.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **Maximum limit of size** Q288: Maximum permissible measured value. Input range 0 to 99999.9999
- **Minimum limit of size** Q289: Minimum permissible measured value. Input range 0 to 99999.9999



NC blocks

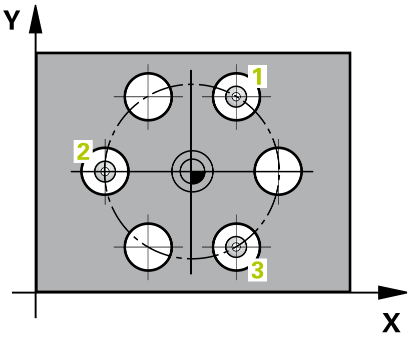
5 TCH PROBE 427 MEASURE COORDINATE	
Q263=+35	**;1ST POINT 1ST AXIS**
Q264=+45	**;1ST POINT 2ND AXIS**
Q261=+5	**;MEASURING HEIGHT**
Q320=0	**;SET-UP CLEARANCE**
Q272=3	**;MEASURING AXIS**
Q267=-1	**;TRAVERSE DIRECTION**
Q260=+20	**;CLEARANCE HEIGHT**
Q281=1	**;MEASURING LOG**
Q288=5.1	**;MAXIMUM DIMENSION**
Q289=4.95	**;MINIMUM DIMENSION**
Q309=0	**;PGM STOP IF ERROR**
Q330=0	**;TOOL**

- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T

16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430)

Cycle run

Touch Probe Cycle 430 finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



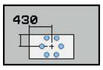
1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the center of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt hole circle diameter

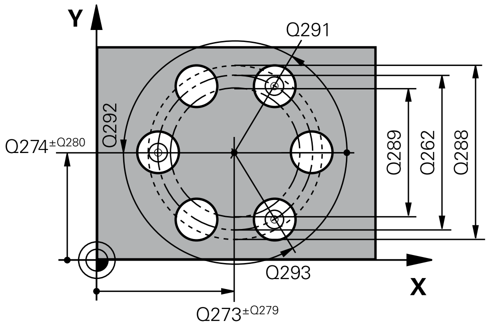
Please note while programming:

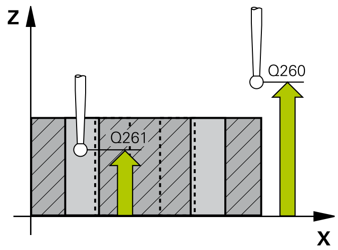
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.
>
> Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

Cycle parameters



- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the bolt hole circle diameter. Input range 0 to 99999.9999
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Maximum limit of size** Q288: Maximum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- **Minimum limit of size** Q289: Minimum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999





NC blocks

```
5 TCH PROBE 430 MEAS. BOLT HOLE
  CIRC
  Q273=+50  ;CENTER IN 1ST AXIS
  Q274=+50  ;CENTER IN 2ND AXIS
  Q262=80   ;NOMINAL DIAMETER
  Q291=+0   ;ANGLE OF 1ST HOLE
  Q292=+90  ;ANGLE OF 2ND HOLE
  Q293=+180;ANGLE OF 3RD HOLE
  Q261=-5   ;MEASURING HEIGHT
  Q260=+10  ;CLEARANCE HEIGHT
  Q288=80.1 ;MAXIMUM DIMENSION
  Q289=79.9 ;MINIMUM DIMENSION
  Q279=0.15 ;TOLERANCE 1ST
            CENTER
```

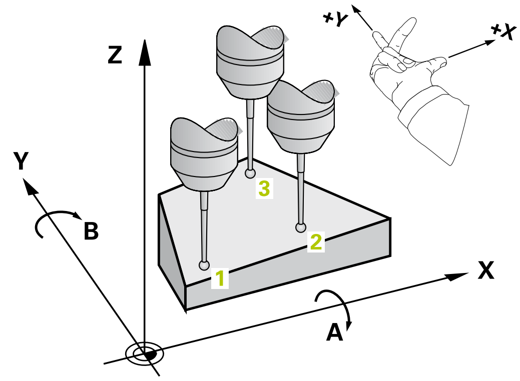
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR430.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor for tool breakage (see "Tool monitoring", page 526). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters.
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T

Q280=0.15	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP IF ERROR
Q330=0	;TOOL

16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)

Cycle run

Touch Probe Cycle 431 finds the angle of a plane by measuring three points. It saves the measured values in system parameters.



1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 444) to the programmed touch point **1** and measures the first point of the plane. The TNC offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
2 The touch probe returns to the clearance height and then moves in the working plane to starting point **2** and measures the actual value of the second touch point of the plane.
3 The touch probe returns to the clearance height and then moves in the working plane to starting point **3** and measures the actual value of the third touch point of the plane.
4 Finally the TNC returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)

Please note while programming:

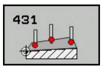
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

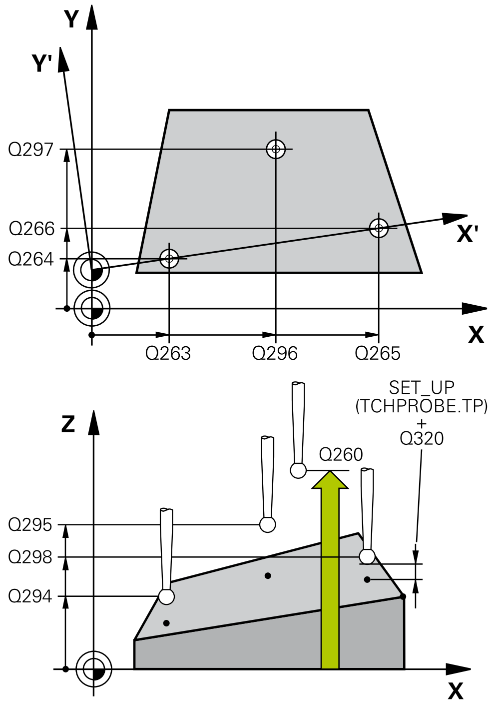
For the TNC to be able to calculate the angular values, the three measuring points must not be positioned on one straight line.

The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.

The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

Cycle parameters





- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 3rd axis** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 3rd axis** Q295 (absolute): Coordinate of the second touch point in the touch probe axis. Input range -99999.9999 to 99999.9999

- **3rd meas. point 1st axis** Q296 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd meas. point 2nd axis** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd meas. point 3rd axis** Q298 (absolute): Coordinate of the third touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the **log file TCHPR431.TXT** as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.

NC blocks

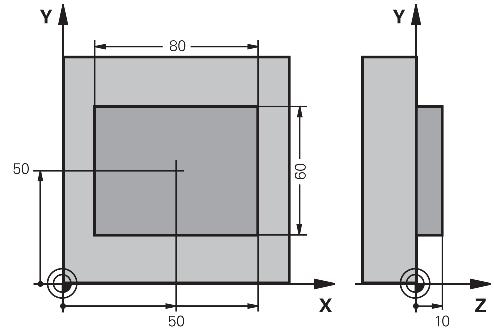
```
5 TCH PROBE 431 MEASURE PLANE
    Q263=+20  ;1ST POINT 1ST AXIS
    Q264=+20  ;1ST POINT 2ND AXIS
    Q294=-10  ;1ST POINT 3RD AXIS
    Q265=+50  ;2ND POINT 1ST AXIS
    Q266=+80  ;2ND POINT 2ND AXIS
    Q295=+0   ;2ND POINT 3RD AXIS
    Q296=+90  ;3RD POINT 1ST AXIS
    Q297=+35  ;3RD POINT 2ND AXIS
    Q298=+12  ;3RD POINT 3RD AXIS
    Q320=0    ;SET-UP CLEARANCE
    Q260=+5   ;CLEARANCE HEIGHT
    Q281=1    ;MEASURING LOG
```

16.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program sequence

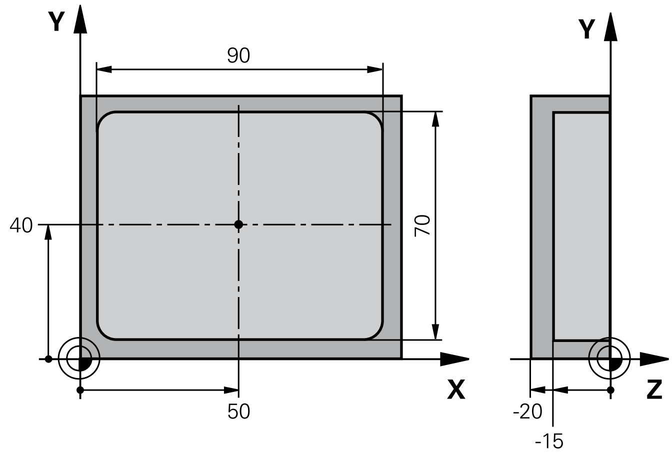
- Roughing with 0.5 mm finishing allowance
- Measuring
- Rectangular stud finishing in accordance with the measured values



0 BEGIN PGM BEAMS MM	
1 TOOL CALL 69 Z	Tool call for roughing
2 L Z+100 R0 FMAX	Retract the tool
3 FN 0: Q1 = +81	Length of rectangle in X (roughing dimension)
4 FN 0: Q2 = +61	Length of rectangle in Y (roughing dimension)
5 CALL LBL 1	Call subprogram for machining
6 L Z+100 R0 FMAX	Retract the tool, change the tool
7 TOOL CALL 99 Z	Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS.	Measure the rough-milled rectangle
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+50 ;CENTER IN 2ND AXIS	
Q282=80 ;FIRST SIDE LENGTH	Nominal length in X (final dimension)
Q283=60 ;2ND SIDE LENGTH	Nominal length in Y (final dimension)
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=0 ;MAX. LIMIT 1ST SIDE	Input values for tolerance checking not required
Q285=0 ;MIN. LIMIT 1ST SIDE	
Q286=0 ;MAX. LIMIT 2ND SIDE	
Q287=0 ;MIN. LIMIT 2ND SIDE	
Q279=0 ;TOLERANCE 1ST CENTER	
Q280=0 ;TOLERANCE 2ND CENTER	
Q281=0 ;MEASURING LOG	No measuring log transmission
Q309=0 ;PGM STOP IF ERROR	Do not output an error message
Q330=0 ;TOOL NO.	No tool monitoring
9 FN 2: Q1 = +Q1 - +Q164	Calculate length in X including the measured deviation
10 FN 2: Q2 = +Q2 - +Q165	Calculate length in Y including the measured deviation
11 L Z+100 R0 FMAX	Retract the touch probe, change the tool

12 TOOL CALL 1 Z S5000	Tool call for finishing
13 CALL LBL 1	Call subprogram for machining
14 L Z+100 R0 FMAX M2	Retract in the tool axis, end program
15 LBL 1	Subprogram with fixed cycle for rectangular stud
16 CYCL DEF 213 STUD FINISHING	
Q200=20 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLING	
Q203=+10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=Q1 ;FIRST SIDE LENGTH	Length in X variable for roughing and finishing
Q219=Q2 ;SECOND SIDE LENGTH	Length in Y variable for roughing and finishing
Q220=0 ;CORNER RADIUS	
Q221=0 ;ALLOWANCE IN 1ST AXS	
17 CYCL CALL M3	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	

Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call for touch probe
2 L Z+100 R0 FMAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+40 ;CENTER IN 2ND AXIS	
Q282=90 ;FIRST SIDE LENGTH	Nominal length in X
Q283=70 ;2ND SIDE LENGTH	Nominal length in Y
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+20 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=90.15 ;MAX. LIMIT 1ST SIDE	Maximum limit in X
Q285=89.95 ;MIN. LIMIT 1ST SIDE	Minimum limit in X
Q286=70.1 ;MAX. LIMIT 2ND SIDE	Maximum limit in Y
Q287=69.9 ;MIN. LIMIT 2ND SIDE	Minimum limit in Y
Q279=0.15 ;TOLERANCE 1ST CENTER	Permissible position deviation in X
Q280=0.1 ;TOLERANCE 2ND CENTER	Permissible position deviation in Y
Q281=1 ;MEASURING LOG	Save measuring log to a file
Q309=0 ;PGM STOP IF ERROR	Do not display an error message in case of a tolerance violation
Q330=0 ;TOOL NUMBER	No tool monitoring
4 L Z+100 R0 FMAX M2	Retract the tool, end program
5 END PGM BSMEAS MM	

17

Touch Probe Cycles: Special Functions

17.1 Fundamentals

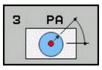
Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

HEIDENHAIN grants a warranty for the function of the touch probe cycles only 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.

The TNC provides a cycle for the following special purpose:

Cycle	Soft key	Page
3 MEASURING Cycle for defining OEM cycles		565

17.2 MEASURE (Cycle 3)

Cycle run

Touch Probe Cycle 3 measures any position on the workpiece in a selectable direction. Unlike other measuring cycles, Cycle 3 enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value after determining the measured value **MB**.

1. The touch probe moves from the current position at the entered feed rate in the defined probing direction. The probing direction must be defined in the cycle as a polar angle.
2. After the TNC has saved the position, the touch probe stops. The TNC saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The TNC does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
3. Finally, the TNC moves the touch probe back by that value against the probing direction that you defined in the parameter **MB**.

Please note while programming:

The exact behavior of Touch Probe Cycle 3 is defined by your machine tool builder or a software manufacturer who uses it within specific touch probe cycles.

The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) data from the touch-probe table, which are effective in other measuring cycles, do not apply in Touch Probe Cycle 3.

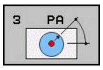
Remember that the TNC always writes to four successive Q-parameters.

If the TNC was not able to determine a valid touch point, the program is run without error message. In this case the TNC assigns the value –1 to the 4th result parameter so that you can deal with the error yourself.

The TNC retracts the touch probe by no more than the retraction distance **MB** and does not pass the starting point of the measurement. This rules out any collision during retraction.

With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.

Cycle parameters



- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters. Input range: 0 to 1999
- **Probing axis**: Enter the axis in whose direction the probe is to move and confirm with the **ENT** key. Input range: X, Y or Z
- **Probing angle**: Angle, measured from the defined **probing axis** in which the touch probe is to move. Confirm with **ENT**. Input range –180.0000 to 180.0000
- **Maximum measuring range**: Enter the maximum distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range -99999.9999 to 99999.9999
- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000
- **Maximum retraction distance**: Traverse path in the direction opposite the probing direction, after the stylus was deflected. The TNC returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input range 0 to 99999.9999
- **Reference system? (0=ACTUAL/1=REF)**: Define whether the probing direction and measuring result should reference the current coordinate system (**ACTUAL**, can be shifted or rotated) or the machine coordinate system (**REF**):
  0: Probe in the current system and save the measuring result to the **ACTUAL** system
  1: Probe in the fixed machine REF system and save the measuring result to the **REF** system.
- **Error mode (0=OFF/1=ON)**: Specify whether the TNC is to issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the TNC saves the value **-1** in the 4th result parameter and continues the cycle:
  0: Error message output
  1: No error message output

NC blocks

```
4 TCH PROBE 3.0 MEASURING
5 TCH PROBE 3.1 Q1
6 TCH PROBE 3.2 X ANGLE: +15
7 TCH PROBE 3.3 DIST +10 F100 MB1
   REFERENCE SYSTEM:0
8 TCH PROBE 3.4 ERRORMODE1
```

17.3 MEASURING IN 3-D (Cycle 4)

Cycle run

Cycle 4 is an auxiliary cycle that can be used for probing with any touch probe (TS, TT or TL). The TNC does not provide a cycle for calibrating the TS touch probe in any probing direction.

Touch probe cycle 4 measures any position on the workpiece in the probing direction defined by a vector. Unlike other measuring cycles, Cycle 4 enables you to enter the measuring distance and feed rate directly. Also, the touch probe retracts by a definable value after determining the measured value.

1. The TNC moves from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).
2. After the TNC has saved the position, the TNC stops the probing motion. The TNC saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.
3. Finally, the TNC performs a positioning movement in the direction opposite to the direction of probing. You define the traverse path in parameter **MB**—the touch probe is moved to a point no farther than the starting point.

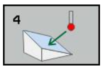
Please note while programming:

The TNC retracts the touch probe by no more than the retraction distance **MB** and does not pass the starting point of the measurement. This rules out any collision during retraction.

Ensure during pre-positioning that the TNC moves the probe-tip center without compensation to the defined position!

Remember that the TNC always writes to four successive Q parameters. If the TNC was not able to determine a valid touch point, the 4th result parameter will have the value –1.

Cycle parameters



- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters. Input range: 0 to 1999
- **Relative measuring path in X**: X component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999
- **Relative measuring path in Y**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999
- **Relative measuring path in Z**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999
- **Maximum measuring path**: Enter the maximum distance from the starting point by which the touch probe may move along the direction vector. Input range -99999.9999 to 99999.9999
- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000
- **Maximum retraction distance**: Traverse path in the direction opposite the probing direction, after the stylus was deflected. Input range 0 to 99999.9999
- **Reference system? (0=ACTUAL/1=REF)**: Define whether the probing result should be saved in the input coordinate system (**ACTUAL**) or referenced to the machine-table coordinate system (**REF**):
  0: Save the measured result in the **ACTUAL** system
  1: Save the measured result in the **REF** system

NC blocks

```
4 TCH PROBE 4.0 MEASURING IN 3-D
5 TCH PROBE 4.1 Q1
6 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
7 TCH PROBE 4.3 DIST+45 F100 MB50
    REFERENCE SYSTEM:0
```

17.4 Calibrating a touch trigger probe

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:

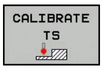
- Commissioning
- Stylus breakage
- Stylus exchange
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

The TNC assumes the calibration values for the active probe system directly after the calibration process. 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.

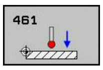
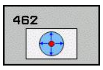
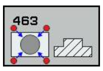
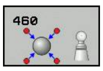
The TNC provides calibration cycles for calibrating the length and the radius:

- Press the **TOUCH PROBE** soft key



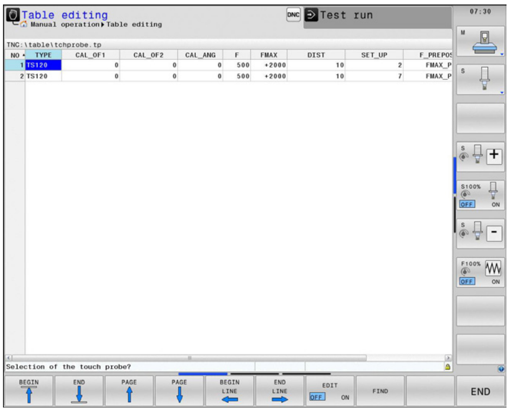
- Display the calibration cycles: Press CALIBRATE TS
- Select the calibration cycle

Calibration cycles of the TNC

Soft key	Function	Page
	Calibrating the length	573
	Measure the radius and the center offset using a calibration ring	575
	Measure the radius and the center offset using a stud or a calibration pin	577
	Measure the radius and the center offset using a calibration sphere	571

17.5 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 offset of the touch probe in the touch-probe table in the **CAL_OF1** (principal axis) and **CAL_OF2** (secondary axis) columns. You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.



A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs. When running a touch probe cycle in the Manual Operation mode, the TNC saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC: \ *.

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.

17.6 CALIBRATE TS (Cycle 460, DIN/ISO: G460)

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically on an exact calibration sphere. You can do radius calibration alone, or radius and length calibration.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

1 Clamp the calibration sphere and check for potential collisions.
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3 The first movement in the cycle is in the negative direction of the touch probe axis.
4 Then the cycle determines the exact center of the sphere in the touch probe axis.

Please note while programming:

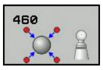
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Pre-position the touch probe in the program so that it is located approximately above the center of the calibration sphere.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.



- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP in the touch probe table. Input range 0 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Number of probe points in the plane (4/3)** Q423: Number of measuring points on the diameter. Input range 0 to 8
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Calibrate length** (0/1) Q433: Define whether the TNC should also calibrate the touch probe length after radius calibration:
  0: Do not calibrate touch probe length
  1: Calibrate touch probe length
- **Datum for length** Q434 (absolute): Coordinate of the calibration sphere center. The definition is only required if length calibration is to be carried out. Input range -99999.9999 to 99999.9999

NC blocks

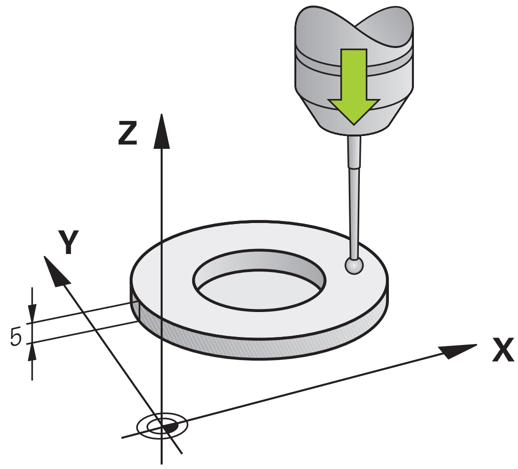
5 TCH PROBE 460 CALIBRATE TS	
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE
Q423=4	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE
Q433=0	;CALIBRATE LENGTH
Q434=-2.5	;DATUM

17.7 CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461)

Cycle run

Before starting the calibration cycle, you must set the datum in the spindle axis so that Z=0 on the machine table; you must also preposition the touch probe over the calibration ring.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

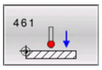


1 The TNC orients the touch probe to the angle **CAL_ANG** from the touch probe table (only if your touch probe can be oriented).
2 The TNC probes from the current position in a negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
3 The TNC then returns the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the start position.

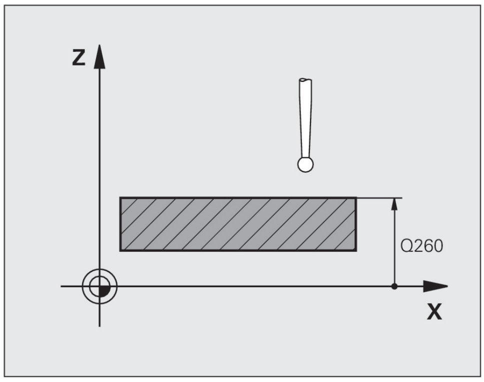
Please note while programming:

⚠	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. Before a cycle definition you must have programmed a tool call to define the touch probe axis. A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.



- **Datum** Q434 (absolute): Datum for the length (e.g. height of the ring gauge). Input range -99999.9999 to 99999.9999



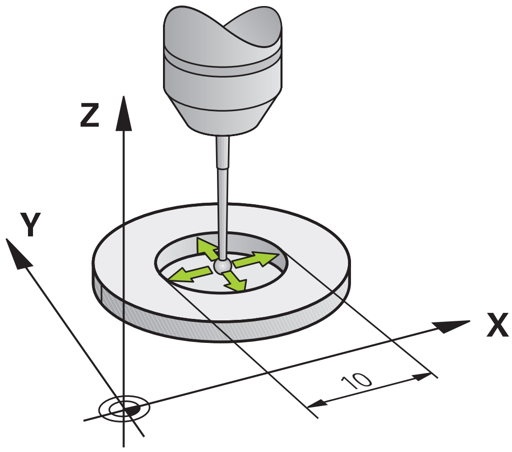
NC blocks

```
5 TCH PROBE 461 CALIBRATE TS
   LENGTH
   Q434=+5    ;DATUM
```

17.8 CALIBRATE TS RADIUS INSIDE (Cycle 462, DIN/ISO: G462)

Cycle run

Before starting the calibration cycle, you need to preposition the touch probe in the center of the calibration ring and at the required measuring height.



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.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The touch probe orientation determines the calibration routine:

- 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 four more probing operations. 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."

Please note while programming:

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

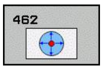
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The center offset can be determined only with a suitable touch probe.

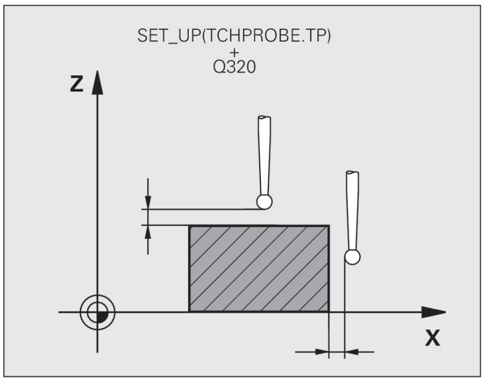
A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information.

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.



- **RING RADIUS** Q407: Diameter of the ring gauge. Input range 0 to 99.9999
- **SET-UP CLEARANCE** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **NO. OF PROBE POINTS** Q407 (absolute): Number of measuring points on the diameter. Input range 0 to 8
- **REFERENCE ANGLE** Q380 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range 0 to 360.0000



NC blocks

5 TCH PROBE 462 TS CALIBRATE IN RING	
Q407=+5	;RING RADIUS
Q320=+0	;SET-UP CLEARANCE
Q423=+8	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE

17.9 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, DIN/ISO: G463)

Cycle run

Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration pin. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration pin.

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.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The touch probe orientation determines the calibration routine:

- 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 four more probing operations. 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."

Please note while programming:

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

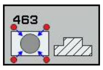
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The center offset can be determined only with a suitable touch probe.

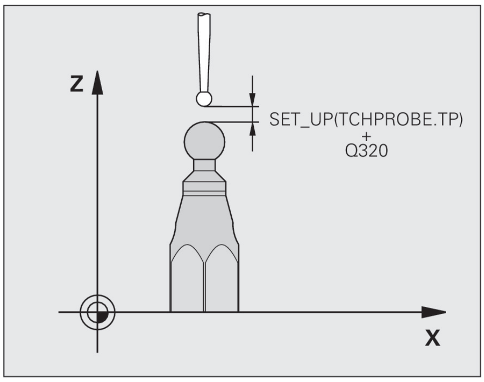
A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information.

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.



- **STUD RADIUS** Q407: Diameter of the ring gauge. Input range 0 to 99.9999
- **SET-UP CLEARANCE** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **MOVE TO CLEARANCE** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **NO. OF PROBE POINTS** Q407 (absolute): Number of measuring points on the diameter. Input range 0 to 8
- **REFERENCE ANGLE** Q380 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range 0 to 360.0000



NC blocks

5 TCH PROBE 463 TS CALIBRATE ON STUD	
Q407=+5	;STUD RADIUS
Q320=+0	;SET-UP CLEARANCE
Q301=+1	;MOVE TO CLEARANCE
Q423=+8	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE

18

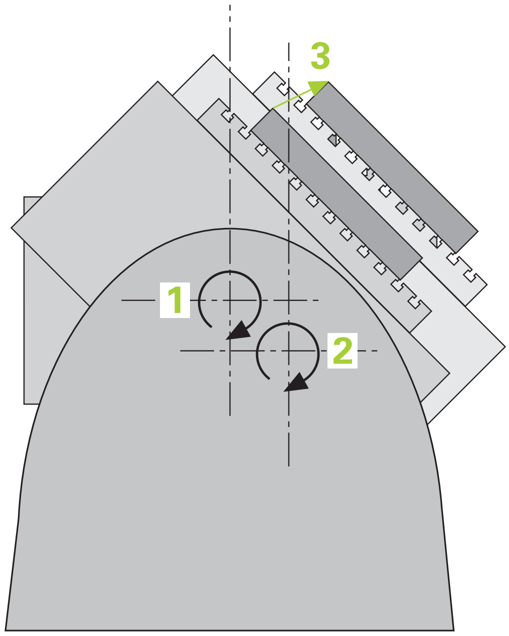
Touch Probe Cycles: Automatic Kinematics Measurement

18.1 Kinematics Measurement with TS Touch Probes (KinematicsOpt option)

Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts need to be manufactured with precision and reproducible accuracy even over long periods.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure at right), and the kinematic conditions actually existing on the machine (see **2** in the figure at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure at right). It is therefore necessary for the model to approach reality as closely as possible.

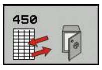


The TNC function **KinematicsOpt** is an important component that helps you to really fulfill these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are in the form of tables or spindle heads. A calibration sphere is fixed at any position on the machine table, and measured with a resolution that you define. During cycle definition you simply define for each rotary axis the area that you want to measure.

From the measured values, the TNC calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematic table.

Overview

The TNC offers cycles that enable you to automatically save, check and optimize the machine kinematics:

Cycle	Soft key	Page
450 SAVE KINEMATICS Automatically saving and restoring kinematic configurations		583
451 MEASURE KINEMATICS Automatically checking or optimizing the machine kinematics		586
452 PRESET COMPENSATION Automatically checking or optimizing the machine kinematics		600

18.2 Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt), 8 (Software option 1) and 17 (Touch Probe function) must be enabled.
- The 3-D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table. HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655 475-01) or **KKH 100 (ID number 655 475-02)**, which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.
- The kinematics description of the machine must be complete and correct. The transformation values must be entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- The machine tool builder must have saved the machine parameters for **CfgKinematicsOpt** in the configuration data. **maxModification** specifies the tolerance limit from which the TNC should indicate if the modifications to kinematic data are above this limit value. **maxDevCalBall** specifies how large the measured calibration sphere radius should be from the cycle parameters entered. **mStrobeRotAxPos** defines an M function specified by the machine tool builder for positioning the rotary axes.

Please note while programming:

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

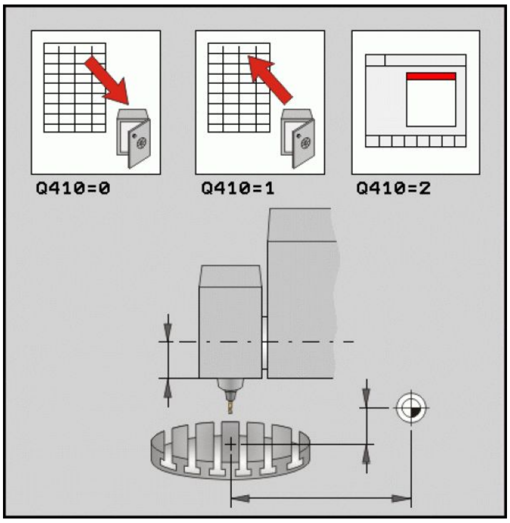
If an M function has been defined in machine parameter **mStrobeRotAxPos**, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450).

If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

18.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option)

Cycle run

With the touch probe cycle 450 you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.



Please note while programming:

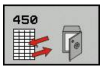
Always save the active kinematics configuration before running a kinematics optimization. Advantage:

- You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).

With the **Restore** mode, note that

- the TNC can restore saved data only to a matching kinematic configuration.
- a change in the kinematics always changes the preset as well. Set the preset again if necessary.

Cycle parameters



- **Mode (0/1/2/3)** Q410: Define if you wish to backup or restore the kinematics:
  0: Backup active kinematics
  1: Restore saved kinematics
  2: Display current memory status
  3: Delete a data record
- **Memory designation** Q409/QS409: Number or name of the data block designator. For a number, enter a value from 0 to 99999; for a name, enter a maximum of 16 characters. 16 memory spaces in total are available. Q409 has no function if Mode 2 has been selected. Wildcards can be used for searching in Modes 1 and 3 (Restore and Delete). If the TNC finds several possible data blocks because of the wildcards, it restores the mean values of the data (Mode 1) or deletes all selected data blocks after confirmation (Mode 3). You can use the following wildcards in a search:
  ?: A single indefinite character
  $: A single alphabetic character (letter)
  #: A single indefinite number
  *****: An indefinite character string of any length

Saving the current kinematics

```
5 TCH PROBE 450 SAVE KINEMATICS
    Q410=0      ;MODE
    QS409="AB";MEMORY DESIGNATION
```

Restoring data blocks

```
5 TCH PROBE 450 SAVE KINEMATICS
    Q410=1      ;MODE
    QS409="AB";MEMORY DESIGNATION
```

Displaying all saved data blocks

```
5 TCH PROBE 450 SAVE KINEMATICS
    Q410=2      ;MODE
    QS409="AB";MEMORY DESIGNATION
```

Deleting data blocks

```
5 TCH PROBE 450 SAVE KINEMATICS
    Q410=3      ;MODE
    QS409="AB";MEMORY DESIGNATION
```

Logging function

After running Cycle 450, the TNC creates a measuring log (**TCHPR450.TXT**) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Save/1=Restore/2=Saving status/3=Delete)
- Designator of the current kinematics
- Entered data record identifier

The other data in the log vary depending on the selected mode:

- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the TNC has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records.
- Mode 3: List of the deleted data records.

Notes on data management

The TNC stores the saved data in the file **TNC:\table\DATA450.KD**. This file can be backed up on an external PC with **TNCREMO**, for example. If the file is deleted, the stored data are removed, too. If the data in the file are changed manually, the data records can become corrupted so that they cannot be used anymore.

If the **TNC:\table\DATA450.KD** file does not exist, it is generated automatically when Cycle 450 is executed.

Do not change stored data manually.

Make a backup of the **TNC:\table\DATA450.KD** file so that you can restore the file, if necessary (e.g. if the data medium is damaged).

18.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

Cycle run

The touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655 475-01) or **KKH 100 (ID number 655 475-02)**, which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The TNC evaluates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

1 Clamp the calibration sphere and check for potential collisions.
2 In the Manual Operation mode, set the reference point in the center of the sphere or, if **Q431=1** or **Q431=3** is defined: Manually position the touch probe over the calibration sphere in the touch probe axis, and in the center of the sphere in the working plane.
3 Select the Program Run mode and start the calibration program.
4 The TNC automatically measures all three axes successively in the resolution you defined.
5 The TNC saves the measured values in the following Q parameters:

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°.

Specify the start and end angles to ensure that the same position is not measured twice. A duplicated point measurement (e.g. measuring positions +90° and –270°) is not advisable, but it does not cause an error message.

- Example: Start angle = +90°, end angle = –90°
  - Start angle = +90°
  - End angle = –90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (–90 – +90) / (4 – 1) = –60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = –30°
  - Measuring point 4 = –90°
- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270 – 90) / (4 – 1) = +60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°

Machines with Hirth-coupled axes

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere. Also ensure that there is enough space to reach the safety clearance (software limit switch).

Define a retraction height **Q408** greater than 0 if software option 2 (**M128**, **FUNCTION TCPM**) is not available.

If necessary, the TNC rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points).

Depending on the machine configuration, the TNC cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine tool builder enabling the TNC to move the rotary axes. The machine tool builder must have entered the number of the M function in machine parameter mStrobeRotAxPos for this purpose.

The measuring positions are calculated from the start angle, end angle and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = –30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (Q412 – Q411) / (Q414 – 1)

Calculated stepping angle = = (90 – –30) / (4 – 1) = 120 / 3 = 40

Measuring position 1 = Q411 + 0 * stepping angle = -30° –> -30°

Measuring position 2 = Q411 + 1 * stepping angle = +10° –> 9°

Measuring position 3 = Q411 + 2 * stepping angle = +50° –> 51°

Measuring position 4 = Q411 + 3 * stepping angle = +90° –> 90°

Choice of number of measuring points

To save time you can make a rough optimization with a small number of measuring points (1 or 2), for example during commissioning.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.

> If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machine with rotary tables/tilting tables: Clamp the calibrating ball as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

Notes on the accuracy

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion of results recorded by the TNC in the measuring log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.

If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.

If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine tool manual provides further information.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere on the machine table so that on rotary table axes there is a large measuring circle, or so that on swivel head axes the measurement can be made at a representative position (e.g. in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made near the rotary axis angle at which the workpiece is to be machined.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is made with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432** you can activate backlash measurement. Enter an angle that the TNC uses as traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the TNC will not measure any backlash.

> The TNC does not perform an automatic backlash compensation.
>
> If the measuring circle radius is < 1 mm, the TNC does not calculate the backlash. The larger the measuring circle radius, the more accurately the TNC can determine the rotary axis backlash (see "Logging function", page 599).
>
> Backlash measurement is not possible if an M function for positioning the rotary axes is set in machine parameter mStrobeRotAxPos or if the axis is a Hirth axis.

Please note while programming:

Note that all functions for tilting in the working plane are reset. **M128** and **FUNCTION TCPM** are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it, or you define the input parameter Q431 correspondingly to 1 or 3.

If machine parameter mStrobeRotAxPos is defined as not equal –1 (M function positions the rotary axis), then only start a measurement when all rotary axes are at 0°.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter **Q253** or the **FMAX** value, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate **Q253**, while the probe monitoring is inactive.

If the kinematic data attained in the Optimize mode are greater than the permissible limit (**maxModification**), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

Note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the TNC first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than you have defined in machine parameter **maxDevCalBall** the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of an emergency the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

The TNC ignores cycle definition data that applies to inactive axes.

Cycle parameters

- **Mode (0=Check/1=Measure)** Q406: Specify whether the TNC should check or optimize the active kinematics:
  0: Check active kinematics. The TNC measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The TNC displays the results of measurement in a measurement log.
  1: Optimize active kinematics. The TNC measures the kinematics in the rotary axes you have defined and **optimizes the position** of the rotary axes of the active kinematics.
- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP in the touch probe table. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the sequence A, then B, then C.
  - Input >0:
    Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. Probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**

Saving and checking the kinematics

```
4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 450 SAVE KINEMATICS
   Q410=0     ;MODE
   Q409=5     ;MEMORY DESIGNATION
6 TCH PROBE 451 MEASURE
   KINEMATICS
   Q406=0     ;MODE
   Q407=12.5  ;SPHERE RADIUS
   Q320=0     ;SET-UP CLEARANCE
   Q408=0     ;RETR. HEIGHT
   Q253=750   ;F PRE-POSITIONING
   Q380=0     ;REFERENCE ANGLE
   Q411=-90   ;START ANGLE A AXIS
   Q412=+90   ;END ANGLE A AXIS
   Q413=0     ;INCID. ANGLE A AXIS
   Q414=0     ;MEAS. POINTS A AXIS
   Q415=-90   ;START ANGLE B AXIS
   Q416=+90   ;END ANGLE B AXIS
   Q417=0     ;INCID. ANGLE B AXIS
   Q418=2     ;MEAS. POINTS B AXIS
   Q419=-90   ;START ANGLE C AXIS
   Q420=+90   ;END ANGLE C AXIS
   Q421=0     ;INCID. ANGLE C AXIS
   Q422=2     ;MEAS. POINTS C AXIS
   Q423=4     ;NO. OF PROBE POINTS
   Q431=0     ;PRESET
   Q432=0     ;BACKLASH, ANG.
              RANGE
```

- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. Input range 0 to 12. If the input value = 0, the TNC does not measure the respective axis.

- **Number meas. points (3-8)** Q423: Number of probe measurements with which the TNC is to measure the calibration sphere in the plane. Input range 3 to 8. Less measuring points increase speed and more measuring points increase measurement precision.
- **Preset (0/1/2/3)** Q431: Define whether the TNC automatically sets the active preset (datum) into the center of the sphere:
  0: Do not set the preset automatically into the center of the sphere: Preset manually before cycle start
  1: Automatically preset into the center of the sphere before measurement: Manually preposition the touch probe above the calibration sphere before the cycle start
  2: Automatically preset into the center of the sphere after measurement: Preset manually before cycle start
  3: Preset before and after measurement into the center of the sphere: Preposition the touch probe manually above the calibration sphere before cycle start
- **Backlash, angle range** Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range -3.0000 to +3.0000

If you have activated "Preset" before the calibration (Q431 = 1/3), then move the touch probe by the safety clearance (Q320 + SET_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.

Various modes (Q406)

Test mode Q406 = 0

- The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The TNC records the results of a possible position optimization but does not make any adjustments.

Position Optimization mode Q406 = 1

- The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the TNC tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data is adjusted automatically.

Position optimization of the rotary axes with preceding, automatic datum setting and measurement of the rotary axis backlash

```
1 TOOL CALL "TCH PROBE" Z
2 TCH PROBE 451 MEASURE
   KINEMATICS
   Q406=1      ;MODE
   Q407=12.5   ;SPHERE RADIUS
   Q320=0      ;SET-UP CLEARANCE
   Q408=0      ;RETR. HEIGHT
   Q253=750    ;F PRE-POSITIONING
   Q380=0      ;REFERENCE ANGLE
   Q411=-90    ;START ANGLE A AXIS
   Q412=+90    ;END ANGLE A AXIS
   Q413=0      ;INCID. ANGLE A AXIS
   Q414=0      ;MEAS. POINTS A AXIS
   Q415=-90    ;START ANGLE B AXIS
   Q416=+90    ;END ANGLE B AXIS
   Q417=0      ;INCID. ANGLE B AXIS
   Q418=0      ;MEAS. POINTS B AXIS
   Q419=+90    ;START ANGLE C AXIS
   Q420=+270   ;END ANGLE C AXIS
   Q421=0      ;INCID. ANGLE C AXIS
   Q422=3      ;MEAS. POINTS C AXIS
   Q423=3      ;NO. OF PROBE POINTS
   Q431=1      ;PRESET
   Q432=0.5    ;BACKLASH, ANG.
               RANGE
```

Logging function

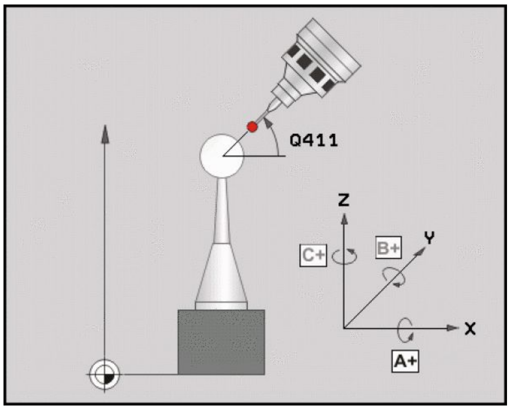
After running Cycle 451, the TNC creates a measuring log **(TCHPR451.TXT)** containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes

18.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

Cycle run

Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)", page 586). Then the TNC corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration sphere after optimization.



This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

1 Clamp the calibration sphere
2 Measure the complete reference head with Cycle 451, and use Cycle 451 to finally set the preset in the center of the sphere.
3 Insert the second head.
4 Use Cycle 452 to measure the interchangeable head up to the point where the head is changed.
5 Use Cycle 452 to adjust other interchangeable heads to the reference head.

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

1 Clamp the calibration sphere and check for potential collisions.
2 Define the preset in the calibration sphere.
3 Set the preset on the workpiece, and start machining the workpiece.
4 Use Cycle 452 for preset compensation at regular intervals. The TNC measures the drift of the axes involved and compensates it in the kinematics description.

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (–1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (–1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Please note while programming:

In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

Note that all functions for tilting in the working plane are reset. **M128** and **FUNCTION TCPM** are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it.

For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse a distance of 1° to the limit switch. The TNC needs this distance for internal backlash compensation.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter **Q253** or the **FMAX** value, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate **Q253**, while the probe monitoring is inactive.

If the kinematic data are greater than the permissible limit (**maxModification**), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

Note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the TNC first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than you have defined in machine parameter **maxDevCalBall** the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

Cycle parameters

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the sequence A, then B, then C.
  - Input >0:
    Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. Probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999

Calibration program

```
4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 450 SAVE KINEMATICS
   Q410=0      ;MODE
   Q409=5      ;MEMORY DESIGNATION
6 TCH PROBE 452 PRESET
   COMPENSATION
   Q407=12.5 ;SPHERE RADIUS
   Q320=0      ;SET-UP CLEARANCE
   Q408=0      ;RETR. HEIGHT
   Q253=750  ;F PRE-POSITIONING
   Q380=0      ;REFERENCE ANGLE
   Q411=-90   ;START ANGLE A AXIS
   Q412=+90  ;END ANGLE A AXIS
   Q413=0      ;INCID. ANGLE A AXIS
   Q414=0      ;MEAS. POINTS A AXIS
   Q415=-90   ;START ANGLE B AXIS
   Q416=+90  ;END ANGLE B AXIS
   Q417=0      ;INCID. ANGLE B AXIS
   Q418=2      ;MEAS. POINTS B AXIS
   Q419=-90   ;START ANGLE C AXIS
   Q420=+90  ;END ANGLE C AXIS
   Q421=0      ;INCID. ANGLE C AXIS
   Q422=2      ;MEAS. POINTS C AXIS
   Q423=4      ;NO. OF PROBE POINTS
   Q432=0      ;BACKLASH, ANG.
               RANGE
```

- **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **No. of measuring points** Q423: Specify the number of probing points to be used by the TNC for measuring the calibration sphere in the plane. Input range: 3 to 8 measurements
- **Backlash, angle range** Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range -3.0000 to +3.0000

Adjustment of interchangeable heads

The goal of this procedure is for the workpiece preset to remain unchanged after changing rotary axes (head exchange).

In the following example, a fork head is adjusted to the A and C axes. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- Insert the interchangeable head that will be used as a reference head.
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle 451 to measure the complete kinematics, including the reference head.
- Set the preset (using Q431 = 2 or 3 in Cycle 451) after measuring the reference head

Measuring a reference head

```
1 TOOL CALL "TCH PROBE" Z
2 TCH PROBE 451 MEASURE
    KINEMATICS
    Q406=1      ;MODE
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=2000   ;F PRE-POSITIONING
    Q380=+45    ;REFERENCE ANGLE
    Q411=-90    ;START ANGLE A AXIS
    Q412=+90    ;END ANGLE A AXIS
    Q413=45     ;INCID. ANGLE A AXIS
    Q414=4      ;MEAS. POINTS A AXIS
    Q415=-90    ;START ANGLE B AXIS
    Q416=+90    ;END ANGLE B AXIS
    Q417=0      ;INCID. ANGLE B AXIS
    Q418=2      ;MEAS. POINTS B AXIS
    Q419=+90    ;START ANGLE C AXIS
    Q420=+270   ;END ANGLE C AXIS
    Q421=0      ;INCID. ANGLE C AXIS
    Q422=3      ;MEAS. POINTS C AXIS
    Q423=4      ;NO. OF PROBE POINTS
    Q431=3      ;PRESET
    Q432=0      ;BACKLASH, ANG.
                RANGE
```

- Insert the second interchangeable head
- Insert the touch probe
- Measure the interchangeable head with Cycle 452
- Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with Q422)
- The preset and the position of the calibration sphere must not be changed during the complete process
- All other interchangeable heads can be adjusted in the same way

The head change function can vary depending on the individual machine tool. Refer to your machine manual.

Adjusting an interchangeable head

```
4 TOOL CALL "TCH PROBE" Z
4 TCH PROBE 452 PRESET
   COMPENSATION
   Q407=12.5 ;SPHERE RADIUS
   Q320=0    ;SET-UP CLEARANCE
   Q408=0    ;RETR. HEIGHT
   Q253=2000;F PRE-POSITIONING
   Q380=+45  ;REFERENCE ANGLE
   Q411=-90  ;START ANGLE A AXIS
   Q412=+90  ;END ANGLE A AXIS
   Q413=45   ;INCID. ANGLE A AXIS
   Q414=4    ;MEAS. POINTS A AXIS
   Q415=-90  ;START ANGLE B AXIS
   Q416=+90  ;END ANGLE B AXIS
   Q417=0    ;INCID. ANGLE B AXIS
   Q418=2    ;MEAS. POINTS B AXIS
   Q419=+90  ;START ANGLE C AXIS
   Q420=+270;END ANGLE C AXIS
   Q421=0    ;INCID. ANGLE C AXIS
   Q422=0    ;MEAS. POINTS C AXIS
   Q423=4    ;NO. OF PROBE POINTS
   Q432=0    ;BACKLASH, ANG.
              RANGE
```

Drift compensation

During machining various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle 452.

- Clamp the calibration sphere
- Insert the touch probe
- Measure the complete kinematics with Cycle 451 before starting the machining process
- Set the preset (using Q432 = 2 or 3 in Cycle 451) after measuring the kinematics.
- Then set the presets on your workpieces and start the machining process

Reference measurement for drift compensation

```
1 TOOL CALL "TCH PROBE" Z
2 CYCL DEF 247 DATUM SETTING
    Q339=1      ;DATUM NUMBER
3 TCH PROBE 451 MEASURE
    KINEMATICS
    Q406=1      ;MODE
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=750    ;F PRE-POSITIONING
    Q380=+45    ;REFERENCE ANGLE
    Q411=+90    ;START ANGLE A AXIS
    Q412=+270   ;END ANGLE A AXIS
    Q413=45     ;INCID. ANGLE A AXIS
    Q414=4      ;MEAS. POINTS A AXIS
    Q415=-90    ;START ANGLE B AXIS
    Q416=+90    ;END ANGLE B AXIS
    Q417=0      ;INCID. ANGLE B AXIS
    Q418=2      ;MEAS. POINTS B AXIS
    Q419=+90    ;START ANGLE C AXIS
    Q420=+270   ;END ANGLE C AXIS
    Q421=0      ;INCID. ANGLE C AXIS
    Q422=3      ;MEAS. POINTS C AXIS
    Q423=4      ;NO. OF PROBE POINTS
    Q431=3      ;PRESET
    Q432=0      ;BACKLASH, ANG.
                RANGE
```

- Measure the drift of the axes at regular intervals.
- Insert the touch probe
- Activate the preset in the calibration sphere.
- Use Cycle 452 to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the complete process

> This procedure can also be performed on machines without rotary axes.

Drift compensation

```
4 TOOL CALL "TCH PROBE" Z
4 TCH PROBE 452 PRESET
   COMPENSATION
   Q407=12.5 ;SPHERE RADIUS
   Q320=0    ;SET-UP CLEARANCE
   Q408=0    ;RETR. HEIGHT
   Q253=99999;F PRE-POSITIONING
   Q380=+45  ;REFERENCE ANGLE
   Q411=-90  ;START ANGLE A AXIS
   Q412=+90  ;END ANGLE A AXIS
   Q413=45   ;INCID. ANGLE A AXIS
   Q414=4    ;MEAS. POINTS A AXIS
   Q415=-90  ;START ANGLE B AXIS
   Q416=+90  ;END ANGLE B AXIS
   Q417=0    ;INCID. ANGLE B AXIS
   Q418=2    ;MEAS. POINTS B AXIS
   Q419=+90  ;START ANGLE C AXIS
   Q420=+270;END ANGLE C AXIS
   Q421=0    ;INCID. ANGLE C AXIS
   Q422=3    ;MEAS. POINTS C AXIS
   Q423=3    ;NO. OF PROBE POINTS
   Q432=0    ;BACKLASH, ANG.
             RANGE
```

Logging function

After running Cycle 452, the TNC creates a measuring log **(TCHPR452.TXT)** containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes

Notes on log data

(see "Logging function", page 599)

19

Touch Probe Cycles: Automatic Tool Measurement

19.1 Fundamentals

Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.

Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

The touch probe cycles are available only with the Touch Probe Functions software option (option number 17). If you are using a HEIDENHAIN touch probe, this option is available automatically.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill
- Tool measurement while the tool is rotating
- Measurement of individual teeth

You can program the cycles for tool measurement in the **Programming** mode of operation using the **TOUCH PROBE** key. The following cycles are available:

Cycle	New format	Old format	Page
Calibrating the TT, Cycles 30 and 480	480 CAL.	30 CAL.	618
Calibrating the wireless TT 449, Cycle 484	484		619
Measuring the tool length, Cycles 31 and 481	481	31	621
Measuring the tool radius, Cycles 32 and 482	482	32	623
Measuring the tool length and radius, Cycles 33 and 483	483	33	625

> The measuring cycles can be used only when the central tool file TOOL.T is active.
>
> Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in controls for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter **Q199**.

Setting machine parameters

> Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgToolMeasurement** and **CfgTTRoundStylus**.
>
> The TNC uses the feed rate for probing defined in **probingFeed** when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius in mm

The feed rate for probing is calculated from:
v = measuring tolerance • n with

v:	Feed rate for probing in mm/min
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc determines the calculation of the probing feed rate:

probingFeedCalc = **ConstantTolerance**:

The measuring tolerance remains constant regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (**maxPeriphSpeedMeas**) and the permissible tolerance (**measureTolerance1**), the sooner you will encounter this effect.

probingFeedCalc = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc = **ConstantFeed**:

The feed rate for probing remains constant; the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance = r • **measureTolerance1**/ 5 mm, where

r:	Active tool radius in mm
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 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 (M3 = -)?**
R_OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L_OFFS	Tool radius 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 0.9999 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?**

Input examples for common tool types

Tool type	CUT	TT:R_OFFS	TT:L_OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill with diameter of < 19 mm	4 (4 teeth)	0 (no offset required because tool diameter is smaller than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from **offsetToolAxis** is used)
End mill with diameter of > 19 mm	4 (4 teeth)	R (offset required because tool diameter is larger than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from **offsetToolAxis** is used)
Radius cutter with a diameter of 10 mm, for example	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured)	5 (always define the tool radius as the offset so that the diameter is not measured in the radius)

19.2 Calibrate the TT (Cycle 30 or 480, DIN/ISO: G480 Touch Probe Functions software option 17)

Cycle run

The TT is calibrated with the measuring cycle TCH PROBE 30 or TCH PROBE 480 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", page 613). The calibration process is automatic. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurements.

Please note while programming:

The functioning of the calibration cycle is dependent on machine parameter **CfgToolMeasurement**. Refer to your machine tool manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The position of the TT within the machine working space must be defined by setting the Machine Parameters **centerPos** > **[0]** to **[2]**.

If you change the setting of any of the Machine Parameters **centerPos** > **[0]** to **[2]**, you must recalibrate.

Cycle parameters

- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999

NC blocks in old format

```
6 TOOL CALL 1 Z
7 TCH PROBE 30.0 CALIBRATE TT
8 TCH PROBE 30.1 HEIGHT: +90
```

NC blocks in new format

```
6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
    Q260=+100;CLEARANCE HEIGHT
```

19.3 Calibrate the wireless TT 449 (Cycle 484, DIN/ISO: G484 Touch Probe Functions)

Fundamentals

With Cycle 484, you can calibrate your tool touch probe, e.g the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run

To calibrate the tool touch probe, program the measuring cycle TCH PROBE 484. In the input parameter Q536, you can specify whether you want to run the cycle semi-automatically or fully automatically.

Semi-automatic—stop before running

- Insert the calibrating tool
- Define and start the calibration cycle
- The TNC interrupts the calibration cycle
- The TNC opens a dialog in a new window
- The dialog asks you to manually position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact

Fully automatic—no stop before running

- Insert the calibrating tool
- Position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact
- Define and start the calibration cycle
- The calibration cycle is executed without stopping. The calibration process starts from the current position of the tool.

Calibrating tool:

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. Enter the exact length and radius of the calibrating tool into the tool table TOOL.T. At the end of the calibration process, the TNC stores the calibration values and takes them into account during subsequent tool measurement. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

Danger of collision!

To avoid collisions, the tool must be pre-positioned before the cycle call if Q536 is set to 1!

In the calibration process, the TNC also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

The functioning of the calibration cycle is dependent on machine parameter **CfgToolMeasurement**. Refer to your machine manual.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. When using a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. The use of a calibrating tool of too small a diameter and/ or protruding too far from the chuck may cause significant inaccuracies.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

Stop before running Q536: Specify whether to stop before cycle start or run the cycle automatically without stopping:
0: Stop before running. A dialog asks you to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC start to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process
1: No stop before running. The TNC starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

NC blocks

```
6 TOOL CALL 1 Z
7 TCH PROBE 484 CALIBRATE TT
    Q536=+0    ;STOP BEFORE RUNNING
```

19.4 Measure the tool length (Cycle 31 or 481, DIN/ISO: G481 Touch Probe Functions software option 17)

Cycle run

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 480 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483"). Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is at standstill.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is at standstill.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**TT: R_OFFS**).

Cycle for measuring a tool during standstill (e.g. for drills)

The control positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until it touches the surface. To activate this function, enter zero for the tool offset: Radius (**TT: R_OFFS**) in the tool table.

Cycle for measuring individual teeth

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. You can enter an additional offset with tool offset: Length (**TT: L_OFFS**) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for CUTTER MEASUREMENT.

Please note while programming:

> Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
>
> You can run an individual tooth measurement of tools with **up to 20 teeth**.

Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q-parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result ?**: Parameter number in which the TNC saves the status of the measurement result:
  0.0: Tool is within tolerance
  1.0: Tool is worn (**LTOL** exceeded)
  2.0: Tool is broken (**LBREAK** exceeded).
  If you do not wish to use the result of measurement within the program, answer the dialog prompt with **NO ENT**.
- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth)

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE
   TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 Q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE
   TEETH:1
```

NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 481 TOOL LENGTH
   Q340=1      ;CHECK
   Q260=+100;CLEARANCE HEIGHT
   Q341=1      ;PROBING THE TEETH
```

19.5 Measure the tool radius (Cycle 32 or 482, DIN/ISO: G482 Touch Probe Functions software option 17)

Cycle run

To measure the tool radius, program the measuring cycle TCH PROBE 32 or TCH PROBE 482 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", page 613). Select via input parameters by which of two methods the radius of a tool is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

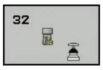
The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control measures the radius of each tooth with the aid of oriented spindle stops.

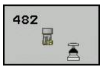
Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgToolMeasurement**. Refer to your machine manual.

Cycle parameters





- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q-parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result ?**: Parameter number in which the TNC saves the status of the measurement result:
  0.0: Tool is within tolerance
  1.0: Tool is worn (**RTOL** exceeded)
  2.0: Tool is broken (**RBREAK** exceeded).
  If you do not wish to use the result of measurement within the program, answer the dialog prompt with **NO ENT**.
- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is also to measure the individual teeth (maximum of 20 teeth)

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE
   TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 Q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE
   TEETH: 1
```

NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 482 TOOL RADIUS
   Q340=1     ;CHECK
   Q260=+100;CLEARANCE HEIGHT
   Q341=1     ;PROBING THE TEETH
```

19.6 Measure the tool length and radius (Cycle 33 or 483, DIN/ISO: G483 Touch Probe Functions software option 17)

Cycle run

To measure both the length and radius of a tool, program the measuring cycle TCH PROBE 33 or TCH PROBE 483 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", page 613). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Via input parameters you can select the desired type of measurement:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for Cycles 31 and 32 as well as .

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgToolMeasurement**. Refer to your machine manual.

Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R and the tool length L in the central tool file TOOL.T by the delta values DR = 0 and DL = 0. If you wish to inspect a tool, the TNC compares the measured data with the tool data stored in TOOL.T. The TNC calculates the deviations and enters them as positive or negative delta values DR and DL in TOOL.T. The deviations are also available in the Q parameters Q115 and Q116. If the delta values are greater than the permissible tool tolerances for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result ?**: Parameter number in which the TNC saves the status of the measurement result:
  0.0: Tool is within tolerance
  1.0: Tool is worn (**LTOL** and/or **RTOL** exceeded)
  2,0: Tool is broken (**LBREAK** and/or **RBREAK** exceeded).
  If you do not wish to use the result of measurement within the program, answer the dialog prompt with **NO ENT**.
- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is also to measure the individual teeth (maximum of 20 teeth)

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 0
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE
   TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 1 Q5
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE
   TEETH: 1
```

NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 483 MEASURE TOOL
   Q340=1     ;CHECK
   Q260=+100;CLEARANCE HEIGHT
   Q341=1     ;PROBING THE TEETH
```

20

Tables of Cycles

20.1 Overview

Fixed cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
7	Datum shift	■		255
8	Mirror image	■		262
9	Dwell time	■		279
10	Rotation	■		264
11	Scaling factor	■		266
12	Program call	■		280
13	Spindle orientation	■		282
14	Contour definition	■		190
19	Tilting the working plane	■		269
20	Contour data SL II	■		195
21	Pilot drilling SL II		■	197
22	Rough out SL II		■	199
23	Floor finishing SL II		■	203
24	Side finishing SL II		■	205
25	Contour train		■	208
270	Contour train data		■	210
26	Axis-specific scaling	■		267
27	Cylinder surface		■	223
28	Cylindrical surface slot		■	226
29	Cylinder surface ridge		■	229
39	Cylinder surface contour		■	232
32	Tolerance	■		283
200	Drilling		■	77
201	Reaming		■	79
202	Boring		■	81
203	Universal drilling		■	84
204	Back boring		■	87
205	Universal pecking		■	90
206	Tapping with a floating tap holder, new		■	105
207	Rigid tapping, new		■	108
208	Bore milling		■	94
209	Tapping with chip breaking		■	111
220	Polar pattern	■		179
221	Cartesian pattern	■		182
225	Engraving		■	300

Cycle number	Cycle designation	DEF active	CALL active	Page
232	Face milling		■	304
233	Face milling (selectable milling direction, consider the side walls)		■	167
240	Centering		■	75
241	Single-lip deep-hole drilling		■	97
247	Datum setting	■		261
251	Rectangular pocket (complete machining)		■	141
252	Circular pocket (complete machining)		■	145
253	Slot milling		■	150
254	Circular slot		■	154
256	Rectangular stud (complete machining)		■	159
257	Circular stud (complete machining)		■	163
262	Thread milling		■	117
263	Thread milling/countersinking		■	120
264	Thread drilling/milling		■	124
265	Helical thread drilling/milling		■	128
267	Outside thread milling		■	132
275	Trochoidal slot		■	211
291	Coupling turning interpolation		■	286
292	Contour turning interpolation		■	295
239	Ascertain the load	■		309

Turning cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
800	Adapt rotary coordinate system	■		322
801	Reset rotary coordinate system	■		328
810	Turn contour, longitudinal		■	344
811	Turn shoulder, longitudinal		■	330
812	Turn shoulder, longitudinal extended		■	333
813	Turn, longitudinal plunge		■	337
814	Turn, longitudinal plunge extended		■	340
815	Turn contour-parallel		■	348
820	Turn contour, transverse		■	366
821	Turn shoulder face		■	352
822	Turn shoulder face extended		■	355
823	Turn, transverse plunge		■	359
824	Turn, transverse plunge extended		■	362
830	Thread, contour-parallel		■	421
831	Thread, longitudinal		■	414
832	Thread, extended		■	417
860	Recessing contour, radial		■	401
861	Recessing, radial		■	394
862	Recessing, radial extended		■	397
870	Recessing contour, axial		■	411
871	Recessing, axial		■	405
872	Recessing, axial extended		■	407
880	Gear hobbing		■	425
892	Check unbalance	■		430

Touch probe cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
0	Reference plane	■		528
1	Polar datum	■		529
3	Measuring	■		565
4	Measuring in 3-D	■		567
30	Calibrate the TT	■		618
31	Measure/Inspect the tool length	■		621
32	Measure/inspect the tool radius	■		623
33	Measure/Inspect the tool length and the tool radius	■		625
400	Basic rotation using two points	■		450
401	Basic rotation over two holes	■		453
402	Basic rotation over two studs	■		456
403	Compensate misalignment with rotary axis	■		459
404	Set basic rotation	■		462
405	Compensate misalignment with the C axis	■		463
408	Reference point at slot center (FCL 3 function)	■		474
409	Reference point at ridge center (FCL 3 function)	■		478
410	Datum from inside of rectangle	■		481
411	Datum from outside of rectangle	■		485
412	Datum from inside of circle (hole)	■		488
413	Datum from outside of circle (stud)	■		493
414	Datum from outside of corner	■		497
415	Datum from inside of corner	■		502
416	Datum from circle center	■		506
417	Datum in touch probe axis	■		510
418	Datum at center between four holes	■		512
419	Datum in any one axis	■		515
420	Workpiece—measure angle	■		530
421	Workpiece—measure hole (center and diameter of hole)	■		533
422	Workpiece—measure circle from outside (diameter of circular stud)	■		536
423	Workpiece—measure rectangle from inside	■		539
424	Workpiece—measure rectangle from outside	■		542
425	Workpiece—measure inside width (slot)	■		545
426	Workpiece—measure outside width (ridge)	■		548
427	Workpiece—measure in any selectable axis	■		551
430	Workpiece—measure bolt hole circle	■		554
431	Workpiece—measure plane	■		554

Cycle number	Cycle designation	DEF active	CALL active	Page
450	KinematicsOpt: Save kinematics (option)	■		583
451	KinematicsOpt: Measure kinematics (option)	■		586
452	KinematicsOpt: Preset compensation	■		580
460	Calibrate the touch probe	■		571
461	Calibrate touch probe length	■		573
462	Calibrate touch probe inside radius	■		575
463	Calibrate touch probe outside radius	■		577
480	Calibrate the TT	■		618
481	Measure/Inspect the tool length	■		621
482	Measure/Inspect the tool radius	■		623
483	Measure/Inspect the tool length and the tool radius	■		625
484	Calibrate TT	■		619

Index

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

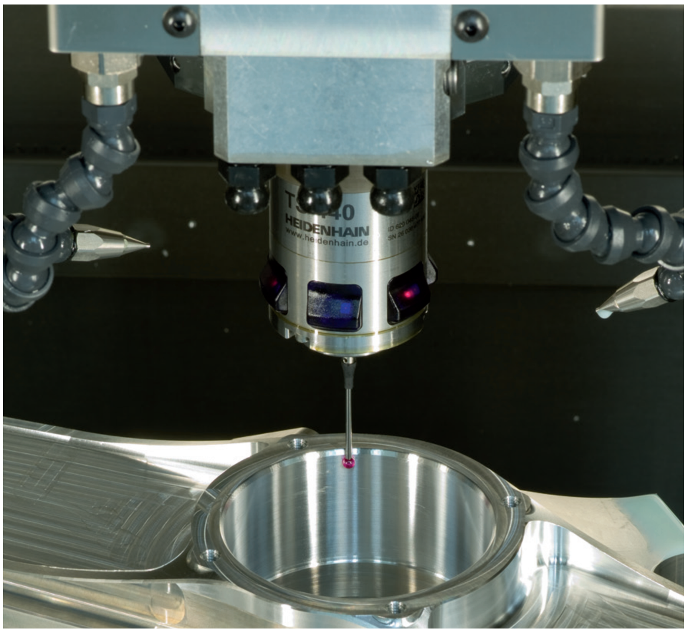
Touch probes from HEIDENHAIN

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

892905-23 · Ver03 · SW05 · 1/2015 · H · Printed in Germany