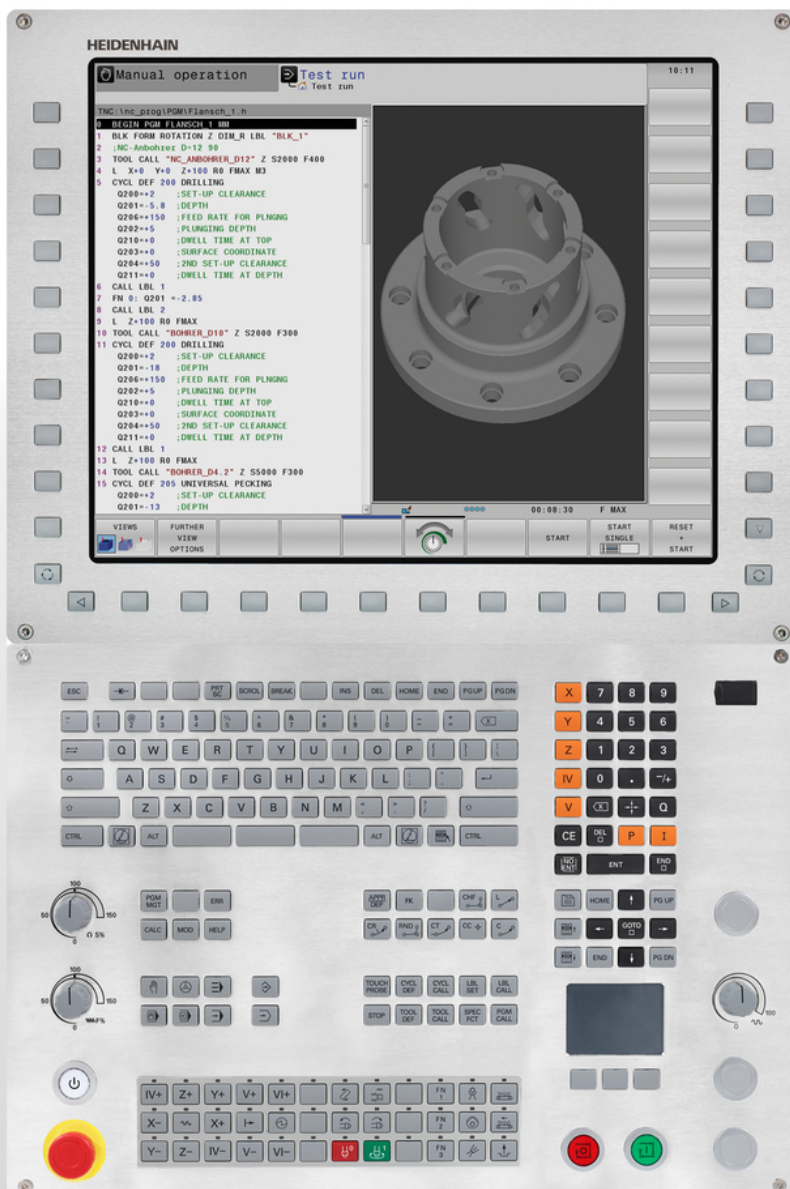




HEIDENHAIN



TNC 640

User's manual for
cycle programming

NC Software
340590-07
340591-07
340595-07

English (en)
9/2016

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	340590-07
TNC 640 E	340591-07
TNC 640 Programming Station	340595-07

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.



Operating instructions:

All TNC functions not connected to the cycles are described in the TNC 640 User's Manual. Please contact HEIDENHAIN if you require a copy of this User's Manual.

ID of User's Manual for conversational programming:
892903-xx.

ID of User's Manual for DIN/ISO programming:
892909-xx.

Software options

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

Additional Axis (options 0 to 7)

Additional axis	Additional control loops 1 to 8
------------------------	---------------------------------

Advanced Function Set 1 (option 8)

Expanded functions Group 1	Machining with rotary tables
-----------------------------------	-------------------------------------

- Cylindrical contours as if in two axes
- Feed rate in distance per minute

Coordinate conversions:

Tilting the working plane

Advanced Function Set 2 (option 9)

Expanded functions Group 2	3-D machining:
-----------------------------------	-----------------------

Export license required

- 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 6 axes

HEIDENHAIN DNC (option 18)

Communication with external PC applications over COM component

Display Step (option 23)

Display step	Input resolution:
---------------------	--------------------------

- Linear axes down to 0.01 µm
- Rotary axes to 0.00001°

Dynamic Collision Monitoring – DCM (option 40)

Dynamic Collision Monitoring	<ul style="list-style-type: none">■ The machine manufacturer defines objects to be monitored■ Warning in Manual operation■ Program interrupt in Automatic operation■ Includes monitoring of 5-axis movements
-------------------------------------	---

DXF Converter (option 42)

DXF converter	<ul style="list-style-type: none">■ Supported DXF format: AC1009 (AutoCAD R12)■ Adoption of contours and point patterns■ Simple and convenient specification of reference points■ Selecting 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
- Cycle 880: Gear hobbing (option 50 and option 131)

KinematicsComp (option 52)

Three-dimensional compensation

Compensation of position and component errors

Export license required

3D-ToolComp (option 92)

3-D tool radius compensation depending on the tool's contact angle

- Compensate the deviation of the tool radius depending on the tool's contact angle
- Compensation values in a separate compensation value table
- Prerequisite: Working with **LN** blocks

Export license required

Extended Tool Management (option 93)

Extended tool management

Python-based

Advanced Spindle Interpolation (option number 96)

Interpolating spindle

Interpolation turning:

- Cycle 291: Interpolation turning, coupling
- Cycle 292: Interpolation turning, contour finishing

Spindle Synchronism (option 131)

Spindle synchronization

- Synchronization of milling spindle and turning spindle
- Cycle 880: Gear hobbing (option 50 and option 131)

Remote Desktop Manager (option 133)

Remote operation of external computer units

- Windows on a separate computer unit
- Incorporated in the TNC interface

Synchronizing Functions (option 135)

Synchronization functions

Real Time Coupling – RTC:

Coupling of axes

Visual Setup Control – VSC (option number 136)

- | | |
|---|--|
| Camera-based monitoring of the setup situation | <ul style="list-style-type: none">■ Record the setup situation with a HEIDENHAIN camera system■ Visual comparison of planned and actual status in the workspace |
|---|--|

Cross Talk Compensation – CTC (option number 141)

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

Position Adaptive Control – PAC (option 142)

- | | |
|----------------------------------|--|
| Adaptive position control | <ul style="list-style-type: none">■ Changing of the control parameters depending on the position of the axes in the working space■ Changing of the control parameters depending on the speed or acceleration of an axis |
|----------------------------------|--|

Load Adaptive Control – LAC (option 143)

- | | |
|------------------------------|--|
| Adaptive load control | <ul style="list-style-type: none">■ 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 number 145)

- | | |
|-------------------------------|---|
| Active chatter control | Fully automatic function for chatter control during machining |
|-------------------------------|---|

Active Vibration Damping – AVD (option number 146)

- | | |
|---------------------------------|--|
| Active vibration damping | Damping of machine oscillations to improve the workpiece surface |
|---------------------------------|--|

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the Feature Content Level (**FCL**) 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 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, which were not all available in some older software versions. Within a cycle, they are always provided at the end of the cycle definition. The section "New and changed cycle functions of software 34059x-05" gives you an overview of the optional Q parameters that have been added in this software version. You can decide for yourself whether you would like to define optional Q 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 318
- New machining cycle 275 Trochoidal milling see "TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)", page 225
- New machining cycle 233 Face milling see "FACE MILLING (Cycle 233, DIN/ISO: G233)", page 180
- In Cycle 205 Universal Pecking you can now use parameter Q208 to define a feed rate for retraction see "Cycle parameters", page 97
- In the thread milling cycles 26x an approaching feed rate was introduced see "Cycle parameters", page 125
- The parameter Q305 NUMBER IN TABLE was added to Cycle 404 see "Cycle parameters", page 486
- 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 97
- 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 102
- The probing cycle 4 MEASURING IN 3-D was introduced see "MEASURING IN 3-D (Cycle 4)", page 595

New and changed cycle functions of software 34059x-05

- New Cycle 880 GEAR HOBBING (software option 50 and software option 131), see "GEAR HOBBING (Cycle 880, DIN/ISO: G880)", page 449
- New Cycle 292 CONTOUR FINISHING TURNING INTERPOLATION (software option 96), see "INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)", page 300
- New Cycle 291 COUPLING TURNING INTERPOLATION (software option 96), see "COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)", page 311
- 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 328
- Cycle 270: CONTOUR TRAIN DATA was added to the cycle package (software option 19), see "CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)", page 224
- 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 246
- 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 318
- Cycles 252 to 254 were expanded by the optional parameter Q439, see "Cycle parameters", page 155
- Cycle 22 was expanded by the optional parameters Q401 and Q404, see "ROUGHING (Cycle 22, DIN/ISO: G122)", page 213
- Cycles 841, 842, 851 and 852 were expanded by the plunging feed rate Q488, see "Cycle parameters", page 392
- Cycle 484 was expanded by the optional parameter Q536, see "Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484)", page 677
- Eccentric turning with Cycle 800 is possible with option 50, see "ADAPT ROTARY COORDINATE SYSTEM(Cycle 800, DIN/ISO: G800)", page 342

New and changed cycle functions of software 34059x-06

- New Cycle 258 POLYGON STUD, see "POLYGON STUD (Cycle 258, DIN/ISO: G258)", page 175
- New Cycles 600 and 601 for camera-based monitoring of the setup situation (software option 136), see "Camera-based monitoring of the setup situation VSC (option number 136)", page 616
- Cycle 291 INTERPOLATION TURNING, COUPLING (software option 96) was expanded to include parameter Q561, see "COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)", page 311
- Cycles 421, 422 and 427 have been expanded to include parameters Q498 and Q531, see "MEASURE HOLE (Cycle 421, DIN/ISO: G421)", page 557
- Cycle 247 DATUM SETTING: The number of the preset can be selected from the preset table, see "DATUM SETTING (Cycle 247, DIN/ISO: G247)", page 275
- Cycles 200 and 203: The behavior of the dwell time at top was modified, see "UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)", page 88
- Cycle 205 performs deburring on the coordinate surface, see "UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)", page 95
- For SL cycles, M110 is now taken into account for arcs compensated on the inside of the arc if M110 is active during machining, see "SL Cycles", page 202

New and changed cycle functions of software 34059x-07

- New Cycle 444 for three-dimensional probing of any coordinate, see "3D PROBING (Cycle 444)", page 597
- Cycle 451 has been expanded by parameter Q406. This makes it possible to use KinematicsComp (software option 52) to compensate the measured angle position errors of the rotary axes, see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)", page 644
- Cycle 460 has been expanded by parameter Q455. This makes it possible to measure, store, and compensate the deviations determined by KinematicsComp (software option 52), see "CALIBRATE TS (Cycle 460, DIN/ISO: G460)", page 604
- The positions of the measured rotary axes before and after optimization can be output in the log of KinematicsOpt Cycles 451 and 452, see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)", page 644, see "PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)", page 658
- Cycle 225 has been expanded by parameters Q516, Q367, and Q574. This makes it possible to define a datum for the respective text position, as well as to scale the text length and character height. The pre-positioning for engraving on a circular path has changed, see "ENGRAVING (Cycle 225, DIN/ISO: G225)", page 318
- Cycle 861 has been expanded by parameters Q510, Q511, and Q462. This makes it possible to program an overlap, a feed rate factor, and a selectable retraction behavior, see "RADIAL RECESSING (Cycle 861, DIN/ISO: G861)", page 414
- Cycle 862 has been expanded by parameters Q510, Q511, and Q462. This makes it possible to program an overlap, a feed rate factor, and a selectable retraction behavior, see "RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)", page 418
- Cycle 871 has been expanded by parameters Q510, Q511, and Q462. This makes it possible to program an overlap, a feed rate factor, and a selectable retraction behavior, see "AXIAL RECESSING (Cycle 871, DIN/ISO: G871)", page 426
- Cycle 872 has been expanded by parameters Q510, Q511, and Q462. This makes it possible to program an overlap, a feed rate factor, and a selectable retraction behavior, see "AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)", page 429

- Cycle 860 has been expanded by parameters Q510, Q511, and Q462. This makes it possible to program an overlap, a feed rate factor, and a selectable retraction behavior, see "RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)", page 422
- Cycle 870 has been expanded by parameters Q510, Q511, and Q462. This makes it possible to program an overlap, a feed rate factor, and a selectable retraction behavior, see "AXIAL RECESSING (Cycle 870, DIN/ISO: G870)", page 434
- In Cycle 810, parameter Q499 was expanded with the input option "2". This modifies the tool position if the contour is machined in the direction opposite of the programmed direction, see "TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)", page 364
- In Cycle 815, parameter Q499 was expanded with the input option "2". This modifies the tool position if the contour is machined in the direction opposite of the programmed direction, see "TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)", page 368
- In Cycle 820, parameter Q499 was expanded with the input option "2". This modifies the tool position if the contour is machined in the direction opposite of the programmed direction, see "TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)", page 386
- In Cycles 481 to 483, parameter Q340 was expanded with the input option "2". This makes it possible to check the tool without changing the tool table, see "Measuring tool length (Cycle 31 or 481, DIN/ISO: G481)", page 679, see "Measuring tool radius (Cycle 32 or 482, DIN/ISO: G482)", page 681, see "Measuring tool length and radius (Cycle 33 or 483, DIN/ISO: G483)", page 683
- Cycle 251 has been expanded by parameter Q439. In addition, the strategy for finishing was revised, see "RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251)", page 147
- In Cycle 252, the strategy for finishing was revised, see "CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)", page 152
- Cycle 275 has been expanded by parameters Q369 and Q439, see "TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)", page 225

Contents

1	Fundamentals / Overviews.....	53
2	Using Fixed Cycles.....	57
3	Fixed Cycles: Drilling.....	77
4	Fixed Cycles: Tapping / Thread Milling.....	109
5	Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling.....	145
6	Fixed Cycles: Pattern Definitions.....	191
7	Fixed Cycles: Contour Pocket.....	201
8	Fixed Cycles: Cylindrical Surface.....	235
9	Fixed Cycles: Contour Pocket with Contour Formula.....	253
10	Cycles: Coordinate Transformations.....	267
11	Cycles: Special Functions.....	291
12	Cycles: Turning.....	335
13	Using Touch Probe Cycles.....	461
14	Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....	471
15	Touch Probe Cycles: Automatic Datum Setting.....	491
16	Touch Probe Cycles: Automatic Workpiece Inspection.....	545
17	Touch Probe Cycles: Special Functions.....	591
18	Visual Setup Control VSC (software option 136).....	615
19	Touch Probe Cycles: Automatic Kinematics Measurement.....	637
20	Touch Probe Cycles: Automatic Tool Measurement.....	669
21	Tables of Cycles.....	685

1	Fundamentals / Overviews.....	53
1.1	Introduction.....	54
1.2	Available Cycle Groups.....	55
	Overview of fixed cycles.....	55
	Overview of touch probe cycles.....	56

2	Using Fixed Cycles.....	57
2.1	Working with fixed cycles.....	58
	Machine-specific cycles.....	58
	Defining a cycle using soft keys.....	59
	Defining a cycle using the GOTO function.....	59
	Calling a cycle.....	60
2.2	Program defaults for cycles.....	62
	Overview.....	62
	Entering GLOBAL DEF.....	62
	Using GLOBAL DEF information.....	63
	Global data valid everywhere.....	64
	Global data for drilling operations.....	64
	Global data for milling operations with pocket cycles 25x.....	64
	Global data for milling operations with contour cycles.....	65
	Global data for positioning behavior.....	65
	Global data for probing functions.....	65
2.3	PATTERN DEF pattern definition.....	66
	Application.....	66
	Entering PATTERN DEF.....	67
	Using PATTERN DEF.....	67
	Defining individual machining positions.....	68
	Defining a single row.....	68
	Defining a single pattern.....	69
	Defining individual frames.....	70
	Defining a full circle.....	71
	Defining a pitch circle.....	72
2.4	Point tables.....	73
	Application.....	73
	Creating a point table.....	73
	Hiding single points from the machining process.....	74
	Selecting a point table in the program.....	74
	Calling a cycle in connection with point tables.....	75

3	Fixed Cycles: Drilling.....	77
3.1	Fundamentals.....	78
	Overview.....	78
3.2	CENTERING (Cycle 240, DIN/ISO: G240).....	79
	Cycle run.....	79
	Please note while programming:.....	79
	Cycle parameters.....	80
3.3	DRILLING (Cycle 200).....	81
	Cycle run.....	81
	Please note while programming:.....	81
	Cycle parameters.....	82
3.4	REAMING (Cycle 201, DIN/ISO: G201).....	83
	Cycle run.....	83
	Please note while programming:.....	83
	Cycle parameters.....	84
3.5	BORING (Cycle 202, DIN/ISO: G202).....	85
	Cycle run.....	85
	Please note while programming:.....	86
	Cycle parameters.....	87
3.6	UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203).....	88
	Cycle run.....	88
	Please note while programming:.....	88
	Cycle parameters.....	89
3.7	BACK BORING (Cycle 204, DIN/ISO: G204).....	91
	Cycle run.....	91
	Please note while programming:.....	92
	Cycle parameters.....	93
3.8	UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205).....	95
	Cycle run.....	95
	Please note while programming:.....	96
	Cycle parameters.....	97

3.9 BORE MILLING (Cycle 208).....	99
Cycle run.....	99
Please note while programming:.....	100
Cycle parameters.....	101
3.10 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241).....	102
Cycle run.....	102
Please note while programming:.....	102
Cycle parameters.....	103
3.11 Programming Examples.....	105
Example: Drilling cycles.....	105
Example: Using drilling cycles in connection with PATTERN DEF.....	106

4	Fixed Cycles: Tapping / Thread Milling	109
4.1	Fundamentals	110
	Overview	110
4.2	TAPPING with a floating tap holder (Cycle 206, DIN/ISO: G206)	111
	Cycle run	111
	Please note while programming:	112
	Cycle parameters	113
4.3	RIGID TAPPING without a floating tap holder (Cycle 207, DIN/ISO: G207)	114
	Cycle run	114
	Please note while programming:	115
	Cycle parameters	116
	Retracting after a program interruption	116
4.4	TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209)	117
	Cycle run	117
	Please note while programming:	118
	Cycle parameters	119
4.5	Fundamentals of Thread Milling	121
	Prerequisites	121
4.6	THREAD MILLING (Cycle 262, DIN/ISO: G262)	123
	Cycle run	123
	Please note while programming:	124
	Cycle parameters	125
4.7	THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263)	127
	Cycle run	127
	Please note while programming:	128
	Cycle parameters	129
4.8	THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264)	131
	Cycle run	131
	Please note while programming:	132
	Cycle parameters	133

4.9	HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265).....	135
	Cycle run.....	135
	Please note while programming:.....	136
	Cycle parameters.....	137
4.10	OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267).....	139
	Cycle run.....	139
	Please note while programming:.....	140
	Cycle parameters.....	141
4.11	Programming Examples.....	143
	Example: Thread milling.....	143

5	Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling.....	145
5.1	Fundamentals.....	146
	Overview.....	146
5.2	RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251).....	147
	Cycle run.....	147
	Please note while programming:.....	148
	Cycle parameters.....	149
5.3	CIRCULAR POCKET (Cycle 252, DIN/ISO: G252).....	152
	Cycle run.....	152
	Please note while programming:.....	154
	Cycle parameters.....	155
5.4	SLOT MILLING (Cycle 253, DIN/ISO: G253).....	157
	Cycle run.....	157
	Please note while programming:.....	158
	Cycle parameters.....	159
5.5	CIRCULAR SLOT (Cycle 254, DIN/ISO: G254).....	162
	Cycle run.....	162
	Please note while programming:.....	163
	Cycle parameters.....	164
5.6	RECTANGULAR STUD (Cycle 256, DIN/ISO: G256).....	167
	Cycle run.....	167
	Please note while programming:.....	168
	Cycle parameters.....	169
5.7	CIRCULAR STUD (Cycle 257, DIN/ISO: G257).....	171
	Cycle run.....	171
	Please note while programming:.....	172
	Cycle parameters.....	173
5.8	POLYGON STUD (Cycle 258, DIN/ISO: G258).....	175
	Cycle run.....	175
	Please note while programming:.....	176
	Cycle parameters.....	177

5.9	FACE MILLING (Cycle 233, DIN/ISO: G233)	180
	Cycle run	180
	Please note while programming	184
	Cycle parameters	185
5.10	Programming Examples	188
	Example: Milling pockets, studs and slots	188

6	Fixed Cycles: Pattern Definitions.....	191
6.1	Fundamentals.....	192
	Overview.....	192
6.2	POLAR PATTERN (Cycle 220, DIN/ISO: G220).....	193
	Cycle run.....	193
	Please note while programming:.....	193
	Cycle parameters.....	194
6.3	LINEAR PATTERN (Cycle 221, DIN/ISO: G221).....	196
	Cycle run.....	196
	Please note while programming:.....	196
	Cycle parameters.....	197
6.4	Programming Examples.....	198
	Example: Polar hole patterns.....	198

7	Fixed Cycles: Contour Pocket	201
7.1	SL Cycles	202
	Fundamentals	202
	Overview	203
7.2	CONTOUR (Cycle 14, DIN/ISO: G37)	204
	Please note while programming:	204
	Cycle parameters	204
7.3	Superimposed contours	205
	Fundamentals	205
	Subprograms: overlapping pockets	205
	Area of inclusion	206
	Area of exclusion	207
	Area of intersection	208
7.4	CONTOUR DATA (Cycle 20, DIN/ISO: G120)	209
	Please note while programming:	209
	Cycle parameters	210
7.5	PILOT DRILLING (Cycle 21, DIN/ISO: G121)	211
	Cycle run	211
	Please note while programming:	212
	Cycle parameters	212
7.6	ROUGHING (Cycle 22, DIN/ISO: G122)	213
	Cycle run	213
	Please note while programming:	214
	Cycle parameters	215
7.7	FLOOR FINISHING (Cycle 23, DIN/ISO: G123)	217
	Cycle run	217
	Please note while programming:	218
	Cycle parameters	218
7.8	SIDE FINISHING (Cycle 24, DIN/ISO: G124)	219
	Cycle run	219
	Please note while programming:	220
	Cycle parameters	221

7.9 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125).....222

Cycle run.....	222
Please note while programming:.....	222
Cycle parameters.....	223

7.10 CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270).....224

Please note while programming:.....	224
Cycle parameters.....	224

7.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275).....225

Cycle run.....	225
Please note while programming:.....	226
Cycle parameters.....	227

7.12 Programming Examples..... 229

Example: Roughing-out and fine-roughing a pocket.....	229
Example: Pilot drilling, roughing-out and finishing overlapping contours.....	231
Example: Contour train.....	233

8	Fixed Cycles: Cylindrical Surface.....	235
8.1	Fundamentals.....	236
	Overview of cylindrical surface cycles.....	236
8.2	CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, software option 1).....	237
	Cycle run.....	237
	Please note while programming:.....	238
	Cycle parameters.....	239
8.3	CYLINDER SURFACE Slot milling (Cycle 28, DIN/ISO: G128, software option 1).....	240
	Cycle run.....	240
	Please note while programming:.....	241
	Cycle parameters.....	242
8.4	CYLINDER SURFACE Ridge milling (Cycle 29, DIN/ISO: G129, software option 1).....	243
	Cycle run.....	243
	Please note while programming:.....	244
	Cycle parameters.....	245
8.5	CYLINDER SURFACE (Cycle 39, DIN/ISO: G139, software option 1).....	246
	Cycle run.....	246
	Please note while programming:.....	247
	Cycle parameters.....	248
8.6	Programming Examples.....	249
	Example: Cylinder surface with Cycle 27.....	249
	Example: Cylinder surface with Cycle 28.....	251

9 Fixed Cycles: Contour Pocket with Contour Formula.....253

9.1 SL cycles with complex contour formula.....254

Fundamentals.....	254
Selecting a program with contour definitions.....	256
Defining contour descriptions.....	256
Entering a complex contour formula.....	257
Superimposed contours.....	258
Contour machining with SL Cycles.....	260
Example: Roughing and finishing superimposed contours with the contour formula.....	261

9.2 SL cycles with simple contour formula.....264

Fundamentals.....	264
Entering a simple contour formula.....	266
Contour machining with SL Cycles.....	266

10 Cycles: Coordinate Transformations.....	267
10.1 Fundamentals.....	268
Overview.....	268
Effectiveness of coordinate transformations.....	268
10.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54).....	269
Effect.....	269
Cycle parameters.....	269
10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53).....	270
Effect.....	270
Please note while programming:.....	271
Cycle parameters.....	271
Selecting a datum table in the part program.....	272
Editing the datum table in the Programming mode of operation.....	272
Configuring a datum table.....	274
Leaving a datum table.....	274
Status displays.....	274
10.4 DATUM SETTING (Cycle 247, DIN/ISO: G247).....	275
Effect.....	275
Please note before programming:.....	275
Cycle parameters.....	275
Status displays.....	275
10.5 MIRRORING (Cycle 8, DIN/ISO: G28).....	276
Effect.....	276
Please note while programming:.....	277
Cycle parameters.....	277
10.6 ROTATION (Cycle 10, DIN/ISO: G73).....	278
Effect.....	278
Please note while programming:.....	279
Cycle parameters.....	279
10.7 SCALING (Cycle 11, DIN/ISO: G72).....	280
Effect.....	280
Cycle parameters.....	280

10.8 AXIS-SPECIFIC SCALING (Cycle 26).....281

Effect.....	281
Please note while programming:.....	281
Cycle parameters.....	282

10.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, software option 1).....283

Effect.....	283
Please note while programming:.....	284
Cycle parameters.....	284
Resetting.....	285
Positioning the axes of rotation.....	285
Position display in a tilted system.....	286
Monitoring of the working space.....	286
Positioning in a tilted coordinate system.....	287
Combining coordinate transformation cycles.....	287
Procedure for working with Cycle 19 WORKING PLANE.....	288

10.10 Programming Examples..... 289

Example: Coordinate transformation cycles.....	289
--	-----

11 Cycles: Special Functions.....	291
11.1 Fundamentals.....	292
Overview.....	292
11.2 DWELL TIME (Cycle 9, DIN/ISO: G04).....	293
Function.....	293
Cycle parameters.....	293
11.3 PROGRAM CALL (Cycle 12, DIN/ISO: G39).....	294
Cycle function.....	294
Please note while programming:.....	294
Cycle parameters.....	295
11.4 SPINDLE ORIENTATION (Cycle 13, DIN/ISO: G36).....	296
Cycle function.....	296
Please note while programming:.....	296
Cycle parameters.....	296
11.5 TOLERANCE (Cycle 32, DIN/ISO: G62).....	297
Cycle function.....	297
Influences of the geometry definition in the CAM system.....	297
Please note while programming:.....	298
Cycle parameters.....	299
11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96).....	300
Cycle run.....	300
Please note while programming:.....	302
Cycle parameters.....	304
Machining variants.....	306
Defining the tool.....	308
11.7 COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96).....	311
Cycle run.....	311
Please note while programming:.....	311
Cycle parameters.....	313
Defining the tool.....	314

11.8 ENGRAVING (Cycle 225, DIN/ISO: G225).....318

Cycle run.....	318
Please note while programming:.....	318
Cycle parameters.....	319
Allowed engraving characters.....	321
Characters that cannot be printed.....	321
Engraving system variables.....	322

11.9 FACE MILLING (Cycle 232, DIN/ISO: G232).....323

Cycle run.....	323
Please note while programming:.....	325
Cycle parameters.....	326

11.10 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143).....328

Cycle run.....	328
Please note while programming:.....	329
Cycle parameters.....	329

11.11 Programming examples.....330

Example: Interpolation Turning Cycle 291.....	330
Example: Interpolation Turning Cycle 292.....	332

12 Cycles: Turning.....	335
12.1 Turning Cycles (software option 50).....	336
Overview.....	336
Working with turning cycles.....	339
Blank form update (FUNCTION TURNDATA).....	340
12.2 ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800).....	342
Application.....	342
Effect.....	345
Please note while programming:.....	345
Cycle parameters.....	346
12.3 RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801).....	348
Please note while programming:.....	348
Effect.....	348
Cycle parameters.....	348
12.4 Fundamentals of Turning Cycles.....	349
12.5 TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811).....	350
Application.....	350
Roughing cycle run.....	350
Finishing cycle run.....	351
Please note while programming:.....	351
Cycle parameters.....	352
12.6 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812).....	353
Application.....	353
Roughing cycle run.....	353
Finishing cycle run.....	354
Please note while programming:.....	354
Cycle parameters.....	355
12.7 TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813).....	357
Application.....	357
Roughing cycle run.....	357
Finishing cycle run.....	358
Please note while programming:.....	358
Cycle parameters.....	359

12.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)..... 360

Application.....	360
Roughing cycle run.....	360
Finishing cycle run.....	361
Please note while programming:.....	361
Cycle parameters.....	362

12.9 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)..... 364

Application.....	364
Roughing cycle run.....	364
Finishing cycle run.....	365
Please note while programming:.....	365
Cycle parameters.....	366

12.10 TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)..... 368

Application.....	368
Roughing cycle run.....	368
Finishing cycle run.....	369
Please note while programming:.....	369
Cycle parameters.....	370

12.11 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)..... 372

Application.....	372
Roughing cycle run.....	372
Finishing cycle run.....	373
Please note while programming:.....	373
Cycle parameters.....	374

12.12 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)..... 375

Application.....	375
Roughing cycle run.....	375
Finishing cycle run.....	376
Please note while programming:.....	376
Cycle parameters.....	377

12.13TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)..... 379

Application.....	379
Roughing cycle run.....	379
Finishing cycle run.....	380
Please note while programming:.....	380
Cycle parameters.....	381

12.14TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)..... 382

Application.....	382
Roughing cycle run.....	382
Finishing cycle run.....	383
Please note while programming:.....	383
Cycle parameters.....	384

12.15TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)..... 386

Application.....	386
Roughing cycle run.....	386
Finishing cycle run.....	387
Please note while programming:.....	387
Cycle parameters.....	388

12.16SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)..... 390

Application.....	390
Roughing cycle run.....	390
Finishing cycle run.....	391
Please note while programming:.....	391
Cycle parameters.....	392

12.17RADIAL RECESSING EXTENDED (Cycle 842, DIN/ISO: G842)..... 393

Application.....	393
Roughing cycle run.....	393
Finishing cycle run.....	394
Please note while programming:.....	394
Cycle parameters.....	395

12.18 RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)..... 398

Application.....	398
Roughing cycle run.....	398
Finishing cycle run.....	399
Please note while programming:.....	399
Cycle parameters.....	400

12.19 SIMPLE AXIAL RECESSING (Cycle 851, DIN/ISO: G851)..... 402

Application.....	402
Roughing cycle run.....	402
Finishing cycle run.....	403
Please note while programming:.....	403
Cycle parameters.....	404

12.20 AXIAL RECESSING EXTENDED (Cycle 852, DIN/ISO: G852)..... 405

Application.....	405
Roughing cycle run.....	405
Finishing cycle run.....	406
Please note while programming:.....	406
Cycle parameters.....	407

12.21 AXIAL RECESSING (Cycle 850, DIN/ISO: G850)..... 410

Application.....	410
Roughing cycle run.....	410
Finishing cycle run.....	411
Please note while programming:.....	411
Cycle parameters.....	412

12.22 RADIAL RECESSING (Cycle 861, DIN/ISO: G861)..... 414

Application.....	414
Roughing cycle run.....	414
Finishing cycle run.....	415
Please note while programming:.....	415
Cycle parameters.....	416

12.23 RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)..... 418

Application.....	418
Roughing cycle run.....	418
Finishing cycle run.....	419
Please note while programming:.....	419
Cycle parameters.....	420

12.24 RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)..... 422

Application.....	422
Roughing cycle run.....	422
Finishing cycle run.....	423
Please note while programming:.....	423
Cycle parameters.....	424

12.25 AXIAL RECESSING (Cycle 871, DIN/ISO: G871)..... 426

Application.....	426
Roughing cycle run.....	426
Finishing cycle run.....	426
Please note while programming:.....	427
Cycle parameters.....	427

12.26 AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)..... 429

Application.....	429
Roughing cycle run.....	429
Finishing cycle run.....	430
Please note while programming:.....	430
Cycle parameters.....	431

12.27 AXIAL RECESSING (Cycle 870, DIN/ISO: G870)..... 434

Application.....	434
Roughing cycle run.....	434
Finishing cycle run.....	435
Please note while programming:.....	435
Cycle parameters.....	436

12.28THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)..... 438

Application.....	438
Cycle run.....	438
Please note while programming:.....	439
Cycle parameters.....	440

12.29THREAD EXTENDED (Cycle 832, DIN/ISO: G832)..... 441

Application.....	441
Cycle run.....	441
Please note while programming:.....	442
Cycle parameters.....	443

12.30CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ISO: G830)..... 445

Application.....	445
Cycle run.....	445
Please note while programming:.....	446
Cycle parameters.....	447

12.31GEAR HOBBING (Cycle 880, DIN/ISO: G880)..... 449

Cycle run.....	449
Please note while programming:.....	450
Cycle parameters.....	451
Direction of rotation depending on the machining side (Q550).....	453

12.32CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)..... 454

Application.....	454
Please note while programming:.....	455
Cycle parameters.....	456

12.33Example program..... 457

Example: Shoulder with recess.....	457
Example: Gear hobbing.....	459

13 Using Touch Probe Cycles.....	461
13.1 General information about touch probe cycles.....	462
Method of function.....	462
Consideration of a basic rotation in the Manual Operation mode.....	462
Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes.....	462
Touch probe cycles for automatic operation.....	463
13.2 Before You Start Working with Touch Probe Cycles.....	465
Maximum traverse to touch point: DIST in touch probe table.....	465
Set-up clearance to touch point: SET_UP in touch probe table.....	465
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table.....	465
Touch trigger probe, probing feed rate: F in touch probe table.....	466
Touch trigger probe, rapid traverse for positioning: FMAX.....	466
Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table.....	466
Executing touch probe cycles.....	467
13.3 Touch probe table.....	468
General information.....	468
Editing touch probe tables.....	468
touch probe data.....	469

14 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....	471
14.1 Fundamentals.....	472
Overview.....	472
Characteristics common to all touch probe cycles for measuring workpiece misalignment.....	473
14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400).....	474
Cycle run.....	474
Please note while programming:.....	474
Cycle parameters.....	475
14.3 BASIC ROTATION over two holes (Cycle 401, DIN/ISO: G401).....	477
Cycle run.....	477
Please note while programming:.....	477
Cycle parameters.....	478
14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402).....	480
Cycle run.....	480
Please note while programming:.....	480
Cycle parameters.....	481
14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403).....	483
Cycle run.....	483
Please note while programming:.....	483
Cycle parameters.....	484
14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404).....	486
Cycle run.....	486
Cycle parameters.....	486
14.7 Compensating workpiece misalignment by rotating the C axis (Cycle 405, DIN/ISO: G405).....	487
Cycle run.....	487
Please note while programming:.....	488
Cycle parameters.....	488
14.8 Example: Determining a basic rotation from two holes.....	490

15 Touch Probe Cycles: Automatic Datum Setting.....	491
15.1 Fundamentals.....	492
Overview.....	492
Characteristics common to all touch probe cycles for datum setting.....	494
15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408).....	496
Cycle run.....	496
Please note while programming:.....	497
Cycle parameters.....	498
15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409).....	500
Cycle run.....	500
Please note while programming:.....	500
Cycle parameters.....	501
15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410).....	503
Cycle run.....	503
Please note while programming:.....	504
Cycle parameters.....	505
15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411).....	507
Cycle run.....	507
Please note while programming:.....	507
Cycle parameters.....	508
15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412).....	510
Cycle run.....	510
Please note while programming:.....	511
Cycle parameters.....	512
15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413).....	515
Cycle run.....	515
Please note while programming:.....	515
Cycle parameters.....	516
15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414).....	519
Cycle run.....	519
Please note while programming:.....	520
Cycle parameters.....	521

15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415).....524

Cycle run.....	524
Please note while programming:.....	525
Cycle parameters.....	526

15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)..... 528

Cycle run.....	528
Please note while programming:.....	529
Cycle parameters.....	530

15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417).....532

Cycle run.....	532
Please note while programming:.....	532
Cycle parameters.....	533

15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418).....534

Cycle run.....	534
Please note while programming:.....	535
Cycle parameters.....	536

15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419).....538

Cycle run.....	538
Please note while programming:.....	538
Cycle parameters.....	539

15.14 Example: Datum setting in center of a circular segment and on top surface of workpiece.....541

15.15 Example: Datum setting on top surface of workpiece and in center of a bolt hole circle.....542

16 Touch Probe Cycles: Automatic Workpiece Inspection.....	545
16.1 Fundamentals.....	546
Overview.....	546
Recording the results of measurement.....	547
Measurement results in Q parameters.....	549
Classification of results.....	549
Tolerance monitoring.....	549
Tool monitoring.....	550
Reference system for measurement results.....	551
16.2 DATUM PLANE (Cycle 0, DIN/ISO: G55).....	552
Cycle run.....	552
Please note while programming:.....	552
Cycle parameters.....	552
16.3 POLAR DATUM PLANE (Cycle 1).....	553
Cycle run.....	553
Please note while programming:.....	553
Cycle parameters.....	553
16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420).....	554
Cycle run.....	554
Please note while programming:.....	554
Cycle parameters.....	555
16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421).....	557
Cycle run.....	557
Please note while programming:.....	558
Cycle parameters.....	559
16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422).....	562
Cycle run.....	562
Please note while programming:.....	563
Cycle parameters.....	564
16.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423).....	567
Cycle run.....	567
Please note while programming:.....	567
Cycle parameters.....	568

16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)..... 570

Cycle run.....	570
Please note while programming:.....	570
Cycle parameters.....	571

16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)..... 573

Cycle run.....	573
Please note while programming:.....	573
Cycle parameters.....	574

16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)..... 576

Cycle run.....	576
Please note while programming:.....	576
Cycle parameters.....	577

16.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)..... 579

Cycle run.....	579
Please note while programming:.....	579
Cycle parameters.....	580

16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430)..... 582

Cycle run.....	582
Please note while programming:.....	583
Cycle parameters.....	583

16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)..... 585

Cycle run.....	585
Please note while programming:.....	586
Cycle parameters.....	586

16.14 Programming Examples..... 588

Example: Measuring and reworking a rectangular stud.....	588
Example: Measuring a rectangular pocket and recording the results.....	590

17 Touch Probe Cycles: Special Functions.....	591
17.1 Fundamentals.....	592
Overview.....	592
17.2 MEASURE (Cycle 3).....	593
Cycle run.....	593
Please note while programming:.....	593
Cycle parameters.....	594
17.3 MEASURING IN 3-D (Cycle 4).....	595
Cycle run.....	595
Please note while programming:.....	595
Cycle parameters.....	596
17.4 3D PROBING (Cycle 444).....	597
Cycle run.....	597
Cycle parameters.....	599
Please note while programming:.....	601
17.5 Calibrating a touch trigger probe.....	602
17.6 Displaying calibration values.....	603
17.7 CALIBRATE TS (Cycle 460, DIN/ISO: G460).....	604
17.8 CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461).....	608
17.9 CALIBRATE TS RADIUS INSIDE (Cycle 462, DIN/ISO: G462).....	610
17.10 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, DIN/ISO: G463).....	612

18 Visual Setup Control VSC (software option 136)..... 615

18.1 Camera-based monitoring of the setup situation VSC (option number136)..... 616

Fundamentals.....	616
Produce live image.....	618
Manage monitoring data.....	619
Overview.....	620
Results of the image evaluation.....	621
Configuration.....	622
Defining the monitoring area.....	624
Possible queries.....	625

18.2 Workspace Global (Cycle 600)..... 626

Application.....	626
Creating reference images.....	626
Monitoring phase.....	629
Please note while programming:.....	629
Cycle parameters.....	630

18.3 Workspace Local (Cycle 601).....631

Application.....	631
Creating reference images.....	631
Monitoring phase.....	634
Please note while programming:.....	634
Cycle parameters.....	635

19 Touch Probe Cycles: Automatic Kinematics Measurement.....	637
19.1 Kinematics measurement with TS touch probes (KinematicsOpt option).....	638
Fundamentals.....	638
Overview.....	639
19.2 Prerequisites.....	640
Please note while programming:.....	640
19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option).....	641
Cycle run.....	641
Please note while programming:.....	641
Cycle parameters.....	642
Logging function.....	642
Notes on data management.....	643
19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option).....	644
Cycle run.....	644
Positioning direction.....	646
Machines with Hirth-coupled axes.....	647
Choice of number of measuring points.....	648
Choice of the calibration sphere position on the machine table.....	649
Notes on the accuracy.....	649
Notes on various calibration methods.....	650
Backlash.....	651
Please note while programming:.....	652
Cycle parameters.....	653
Various modes (Q406).....	656
Logging function.....	657
19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option).....	658
Cycle run.....	658
Please note while programming:.....	660
Cycle parameters.....	661
Adjustment of interchangeable heads.....	663
Drift compensation.....	665
Logging function.....	667

20 Touch Probe Cycles: Automatic Tool Measurement.....669

20.1 Fundamentals..... 670

Overview.....	670
Differences between Cycles 31 to 33 and Cycles 481 to 483.....	671
Setting machine parameters.....	672
Entries in the tool table TOOL.T.....	674

20.2 Calibrate the TT (Cycle 30 or 480, DIN/ISO: G480 Option 17)..... 676

Cycle run.....	676
Please note while programming:.....	676
Cycle parameters.....	676

20.3 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484).....677

Fundamentals.....	677
Cycle run.....	677
Please note while programming:.....	678
Cycle parameters.....	678

20.4 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481)..... 679

Cycle run.....	679
Please note while programming:.....	680
Cycle parameters.....	680

20.5 Measuring tool radius (Cycle 32 or 482, DIN/ISO: G482).....681

Cycle run.....	681
Please note while programming:.....	681
Cycle parameters.....	682

20.6 Measuring tool length and radius (Cycle 33 or 483, DIN/ISO: G483).....683

Cycle run.....	683
Please note while programming:.....	683
Cycle parameters.....	684

21 Tables of Cycles..... 685

21.1 Overview..... 686

Fixed cycles.....686

Turning cycles.....688

Touch probe cycles.....689

1

**Fundamentals /
Overviews**

1.1 Introduction

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

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

Soft key	Cycle group	Page
	Cycles for pecking, reaming, boring and counterboring	78
	Cycles for tapping, thread cutting and thread milling	110
	Cycles for milling pockets, studs and slots and for face milling	146
	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	268
	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	236
	Cycles for producing point patterns, such as circular or linear hole patterns	192
	Cycles for turning and gear hobbing	336
	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, determining the load	292

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

Overview of touch probe cycles



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

Soft key	Cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	472
	Cycles for automatic workpiece presetting	492
	Cycles for automatic workpiece inspection	546
	Special cycles	592
	Touch probe calibration	604
	Cycles for automatic kinematics measurement	472
	Cycles for automatic tool measurement (enabled by the machine tool builder)	670
	Cycles for VSC: camera-based monitoring of the setup situation (software option 136)	616



- 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. For parallel use of DEF active cycles (cycles that the TNC is automatically running during cycle definition) and CALL active cycles (cycles that you need to call up to run).

Further Information: "Calling a cycle", page 60

Adhere to the following procedure in order to avoid problems regarding the overwriting of transfer parameters that are used more than once:

- ▶ 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



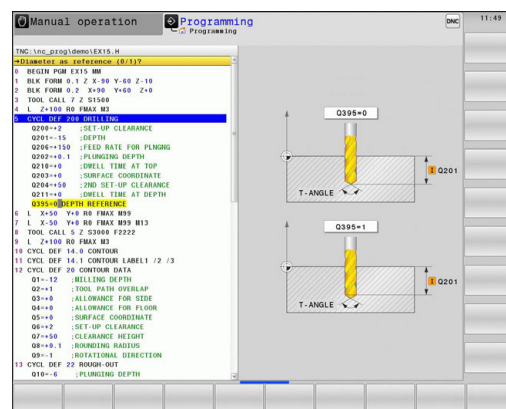
- ▶ The soft-key row shows the available groups of cycles



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



- ▶ 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



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



- ▶ 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
Q210=0	;DWELL TIME AT TOP
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q211=0.25	;DWELL TIME AT DEPTH
Q395=0	;DEPTH REFERENCE

2 Using Fixed Cycles

2.1 Working with fixed cycles

Calling a cycle



Requirements

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 point patterns on circles and Cycle 221 for point patterns on lines
- SL Cycle 14 CONTOUR GEOMETRY
- SL Cycle 20 CONTOUR DATA
- Cycle 32 TOLERANCE
- Coordinate transformation cycles
- Cycle 9 DWELL TIME
- All 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.



- ▶ 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 machining cycle at all positions that you defined in a PATTERN DEF pattern definition or in a points table.

Further Information: "PATTERN DEF pattern definition", page 66

Further Information: "Point tables", page 73

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 for traverse to the start position programmed in this block.

Using positioning logic the TNC moves 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.

Cycle call with M99/M89

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

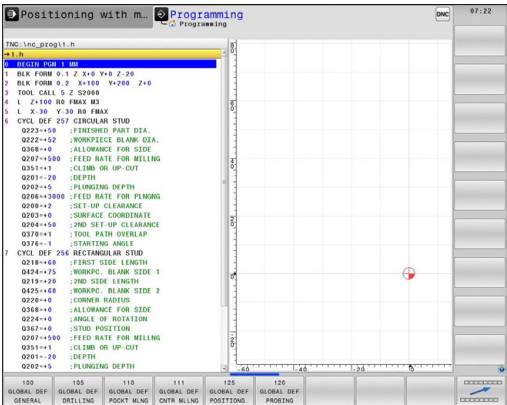
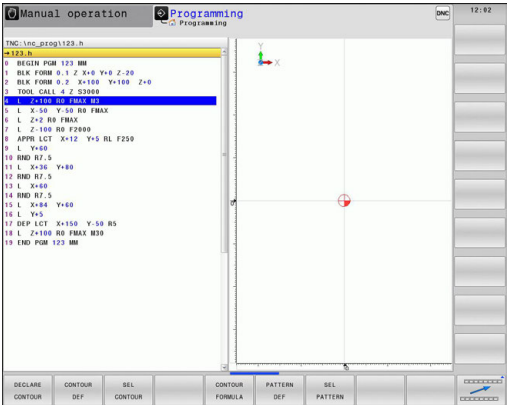
All Cycles 20 to 25, as well as all of those with numbers 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. The **GLOBAL DEF** function gives you the possibility of defining these cycle parameters at the beginning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

The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	64
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	64
110 GLOBAL DEF POCKET MILLING	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	64
111 GLOBAL DEF CNTR MILLING	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	65
125 GLOBAL DEF POSITIONING	GLOBAL DEF POSITIONING Definition of the positioning behavior for CYCL CALL PAT	65
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	65

Entering GLOBAL DEF

- 
 - ▶ Operating mode: Press the **Programming** key
- 
 - ▶ Press the SPEC FCT key to select the special functions
- 
 - ▶ Select the functions for program defaults
- 
 - ▶ Press the **GLOBAL DEF** soft key
- 
 - ▶ Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
 - ▶ Enter the required definitions, and confirm each entry with the **ENT** key

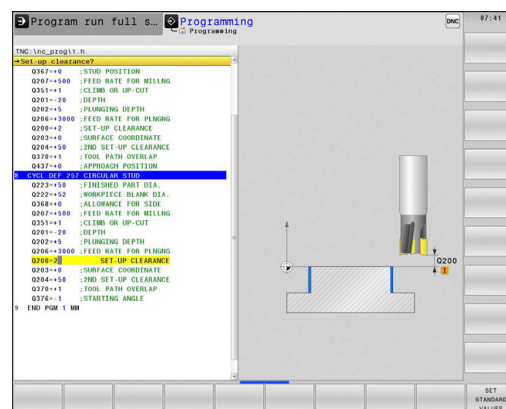


Using GLOBAL DEF information

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:

- ▶ Operating mode: Press the Programming key
- ▶ Select machining cycles: Press the CYCLE DEF key
- ▶ 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.

2 Using Fixed Cycles

2.2 Program defaults for cycles

Global data valid everywhere

- ▶ **Safety 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, 241 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 face 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



The parameters apply to all touch probe cycles with numbers greater than 4xx.

2.3 PATTERN DEF pattern definition

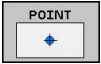

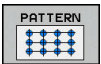
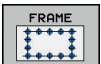


Application

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:

Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	68
	ROW Definition of a single row, straight or rotated	68
	PATTERN Definition of a single pattern, straight, rotated or distorted	69
	FRAME Definition of a single frame, straight, rotated or distorted	70
	CIRCLE Definition of a full circle	71
	PITCH CIRCLE Definition of a pitch circle	72

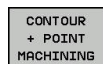
Entering PATTERN DEF



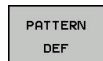
- ▶ Operating mode: Press the **Programming** key



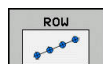
- ▶ Press the SPEC FCT key to select the special functions



- ▶ Select the functions for contour and point machining



- ▶ Press the **PATTERN DEF** soft key



- ▶ Select the desired machining pattern, e.g. press the "single row" soft key
- ▶ 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.

Further Information: "Calling a cycle", page 60

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.

More information: User's Manual for conversational programming

The TNC retracts the tool to the clearance height 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.

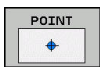
Defining individual machining positions



You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.

POS1 must be programmed with absolute coordinates. POS2 to POS9 can be programmed as absolute and/or incremental values.

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.



- ▶ POS1: **X coord. of machining position** (absolute): Enter the X coordinate
- ▶ POS1: **Y coord. of machining position** (absolute): Enter the Y coordinate
- ▶ POS1: **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin
- ▶ POS2: **X coord. of machining position** (absolute or incremental): Enter the X coordinate
- ▶ POS2: **Y coord. of machining position** (absolute or incremental): Enter Y coordinate
- ▶ POS2: **Z coord. of machining position** (absolute or incremental): Enter Z coordinate

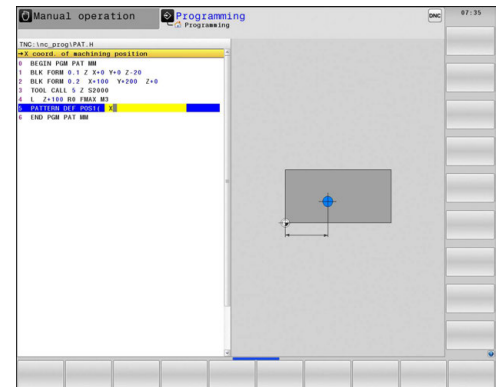
NC blocks

10 L Z+100 R0 FMAX

11 PATTERN DEF

POS1 (X+25 Y+33.5 Z+0)

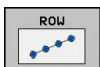
POS2 (X+15 Y+6.5 Z+0)



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.



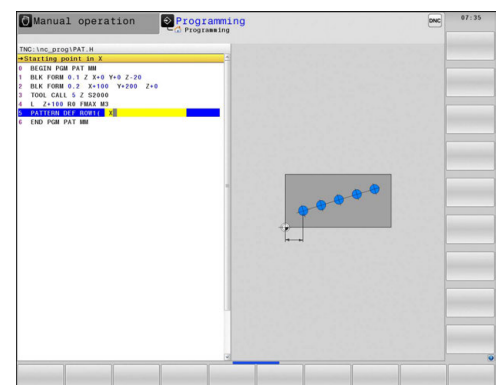
- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the row in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the row in the Y axis
- ▶ **Spacing of machining positions** (incremental): Distance between the machining positions. You can enter a positive or negative value
- ▶ **Number of operations**: Total number of machining positions
- ▶ **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)

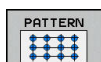


Defining a single pattern



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.** und **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.

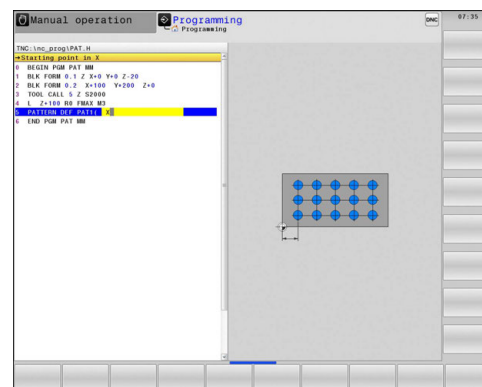


- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions 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 rows**: 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 PAT1 (X+25 Y+33,5
DX+8 DY+10 NUMX5 NUMY4 ROT+0
ROTX+0 ROTY+0 Z+0)

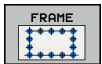


Defining individual frames



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.** und **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.

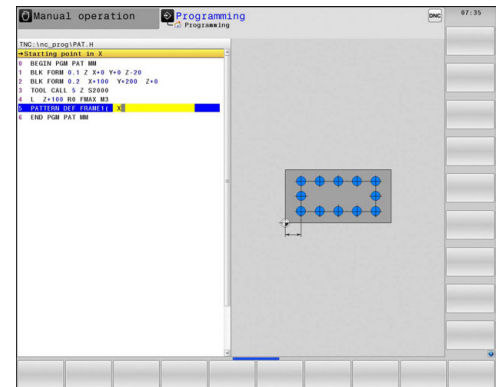


- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the frame in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the frame in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions in the 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 rows**: 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

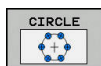
11 PATTERN DEF FRAME1
(X+25 Y+33,5 DX+8 DY+10 NUMX5
NUMY4 ROT+0 ROTX+0 ROTY+0 Z
+0)



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.

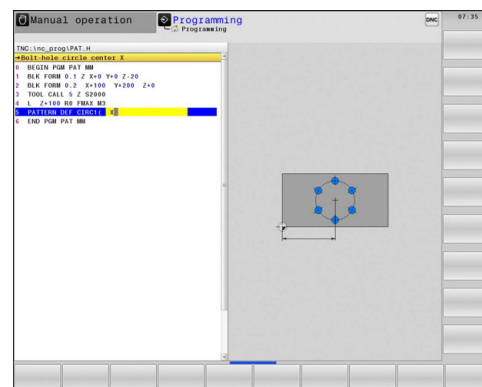


- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis.
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis.
- ▶ **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- ▶ **Starting angle**: Polar angle of the first 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 operations**: 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.

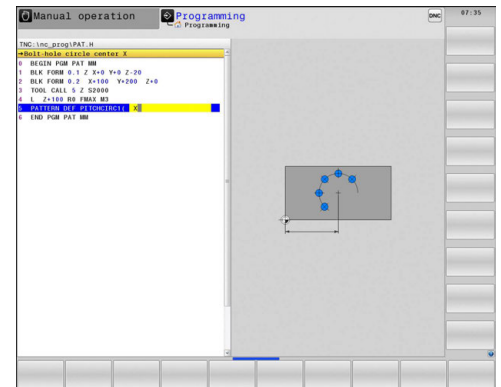


- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis.
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis.
- ▶ **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- ▶ **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- ▶ **Stepping angle/Stopping 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 operations**: 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



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



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

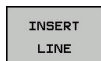
FILE NAME?



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



- ▶ 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.



- ▶ 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



- Activate hiding, or



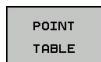
- Deactivate hiding

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:



- Press the **PGM CALL** key to call the function for selecting the 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\NUST35.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**:



- ▶ 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 clearance height 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 table with SL cycles and Cycle 12

The TNC interprets the points as an additional datum shift.

Effect of the point table 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 table 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**


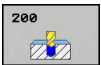
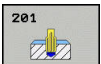
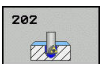
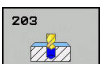
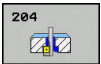



Fixed Cycles: Drilling

3.1 Fundamentals

3.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth	79
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	81
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	83
	202 BORING With automatic pre-positioning, 2nd set-up clearance	85
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	88
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	91
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	95
	208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	99
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	102

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 the 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!

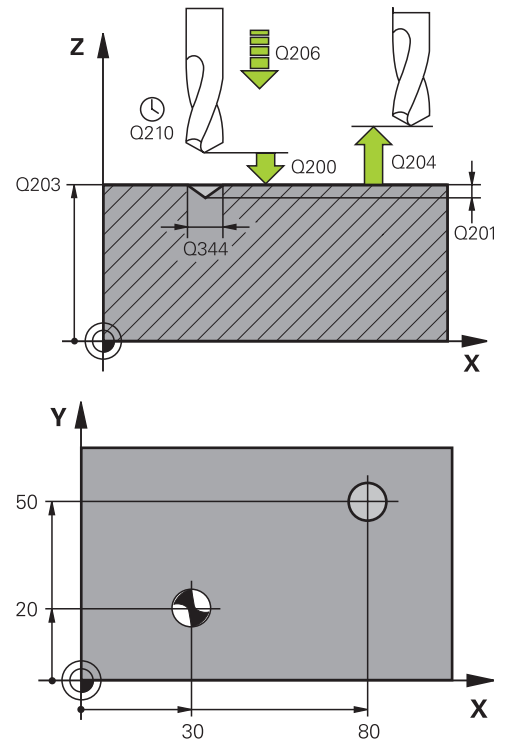
Fixed Cycles: Drilling

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- ▶ **Q343 Select diameter/depth (1/0):** 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
- ▶ **Q201 Depth?** (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
- ▶ **Q344 Diameter of counterbore** (algebraic sign): Centering diameter. Only effective if Q343=1 is defined. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during centering. Input range 0 to 99999.999, alternatively **FAUTO, FU**
- ▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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 DIA./DEPTH	
	Q201=+0		;DEPTH	
	Q344=-9		;DIAMETER	
	Q206=250		;FEED RATE FOR PLNGNG	
	Q211=0.1		;DWELL TIME AT DEPTH	
	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!

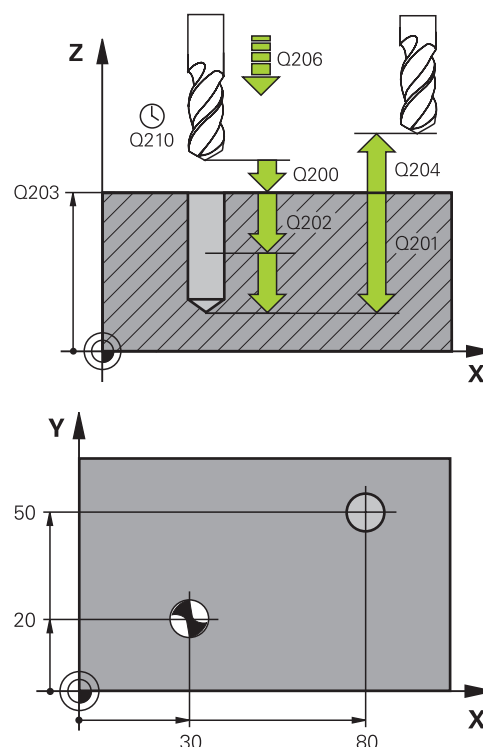
Fixed Cycles: Drilling

3.3 DRILLING (Cycle 200)

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- ▶ **Q202 Plunging depth?** (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
- ▶ **Q210 Dwell time at the top?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 3600.0000
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q395 Diameter as reference (0/1)?**: 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 DEPTH
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_r 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!

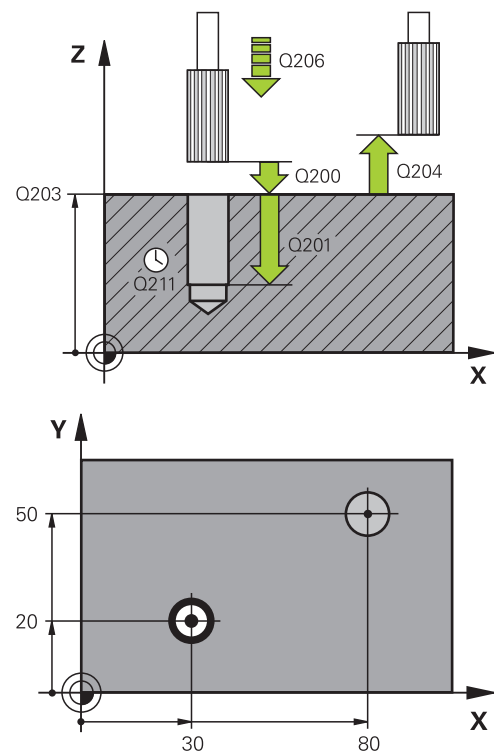
Fixed Cycles: Drilling

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during reaming. Input range 0 to 99999.999, alternatively **FAUTO, FU**
- ▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for reaming applies. Input range 0 to 99999.999
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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 DEPTH
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.
- 7 The TNC finally positions the tool back at the center of the hole.

Fixed Cycles: Drilling

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

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 machining, the TNC positions the tool back at the starting point of the machining plane. This way, you can continue positioning incrementally.

If the functions M7 or M8 were active before calling the cycle, the TNC will reconstruct this previous state at the end of 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).

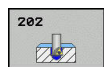
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

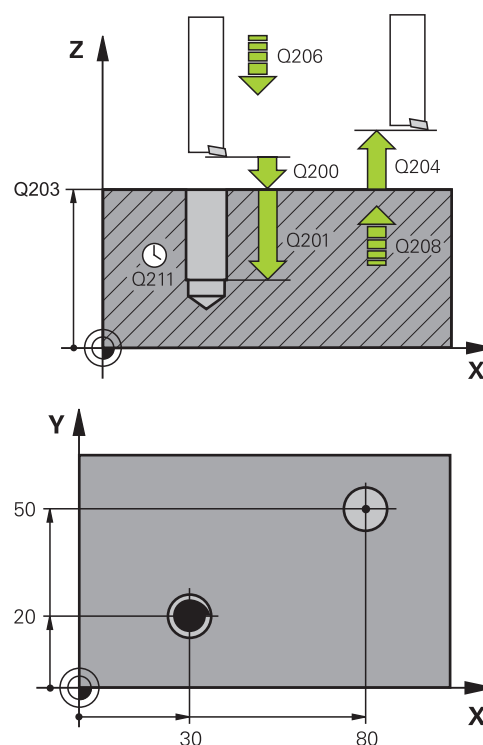
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the **Positioning with manl.data input** mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during boring. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- ▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for plunging applies. Input range 0 to 99999.999, alternatively **FMAX**, **FAUTO**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q214 Disengaging directn (0/1/2/3/4)?**: 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 negative direction of the principle axis
 - 2**: Retract the tool in negative direction of the minor axis
 - 3**: Retract the tool in positive direction of the principle axis
 - 4**: Retract the tool in positive direction of the minor axis
- ▶ **Q336 Angle for spindle orientation?** (absolute): Angle to 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 DEPTH
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

Fixed Cycles: Drilling

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

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!

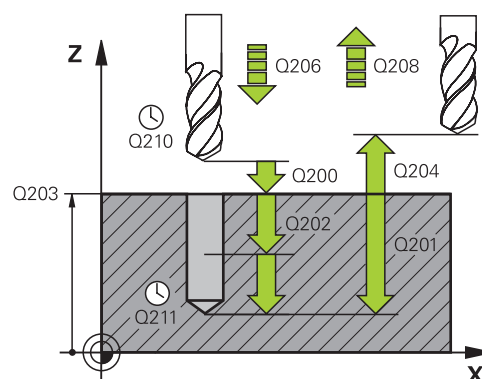
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off). Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO, FU**
- ▶ **Q202 Plunging depth?** (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
- ▶ **Q210 Dwell time at the top?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 3600.0000
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q212 Decrement?** (incremental): Value by which the TNC decreases **Q202 MAX. PLUNGING DEPTH** after each infeed. Input range 0 to 99999.9999
- ▶ **Q213 Nr of breaks before retracting?**: 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
- ▶ **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the TNC limits the plunging depth to the value for **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	;NR OF BREAKS
Q205=3	;MIN. PLUNGING DEPTH
Q211=0.25	;DWELL TIME AT DEPTH
Q208=500	;RETRACTION FEED RATE
Q256=0.2	;DIST FOR CHIP BRKNG
Q395=0	;DEPTH REFERENCE

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

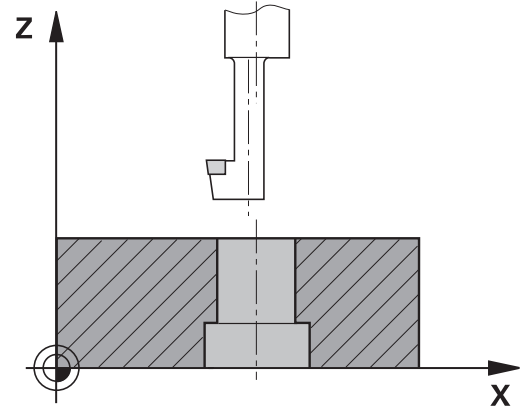
- ▶ **Q211 Dwell time at the depth?:** Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q208 Feed rate for retraction?:** Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.999, alternatively **FMAX**, **FAUTO**
- ▶ **Q256 Retract dist. for chip breaking?**
(incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999
- ▶ **Q395 Diameter as reference (0/1)?:** 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**.
- 7 The TNC finally positions the tool back at the center of the hole.



Fixed Cycles: Drilling

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

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**.

After machining, the TNC positions the tool back at the starting point of the machining plane. This way, you can continue positioning incrementally.

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.

If the functions M7 or M8 were active before calling the cycle, the TNC will reconstruct this previous state at the end of the cycle.



Danger of collision!

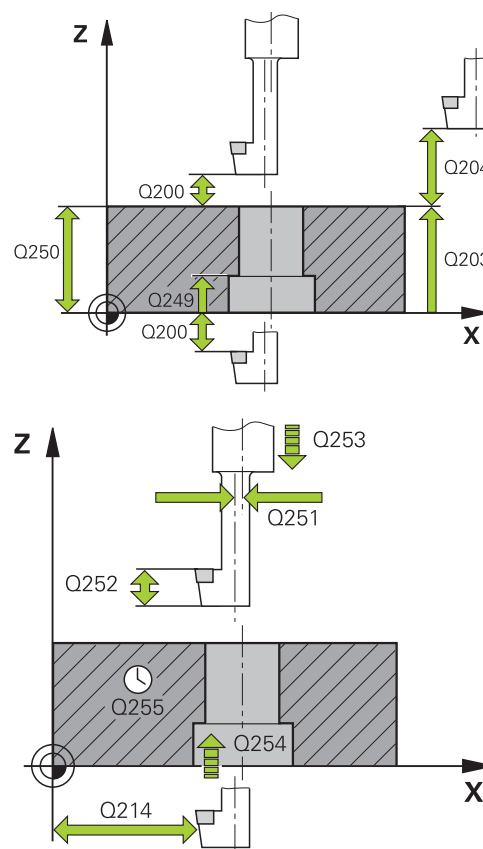
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in **Q336** (for example, in the **Positioning with manl.data input** mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which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

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q249 Depth of counterbore?** (incremental): Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range -99999.9999 to 99999.9999
- ▶ **Q250 Material thickness?** (incremental): Thickness of the workpiece Input range 0.0001 to 99999.9999
- ▶ **Q251 Tool edge off-center distance?** (incremental): Off-center distance for the boring bar; value from the tool data sheet Input range 0.0001 to 99999.9999
- ▶ **Q252 Tool edge height?** (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
- ▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- ▶ **Q255 Dwell time in secs.?**: Dwell time in seconds at the top of the bore hole. Input range 0 to 3600.000
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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 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

Fixed Cycles: Drilling

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

- ▶ **Q214 Disengaging directn (0/1/2/3/4)?**: 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 negative direction of the principle axis
 - 2**: Retract the tool in negative direction of the minor axis
 - 3**: Retract the tool in positive direction of the principle axis
 - 4**: Retract the tool in positive direction of the minor axis
- ▶ **Q336 Angle for spindle orientation?** (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

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**.

Fixed Cycles: Drilling

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

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. The TNC does not change retracting movements; they are referenced 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).

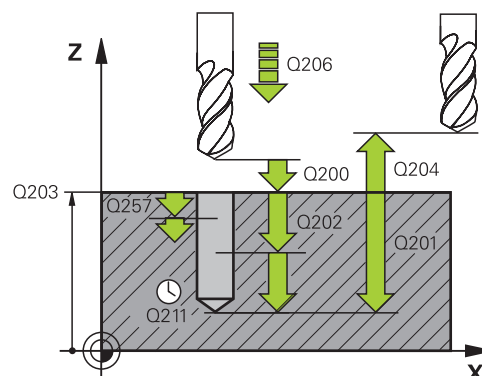
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO, FU**
- ▶ **Q202 Plunging depth?** (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
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q212 Decrement?** (incremental): Value by which the TNC decreases the plunging depth Q202. Input range 0 to 99999.9999
- ▶ **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the TNC limits the plunging depth to the value for **Q205**. Input range 0 to 99999.9999
- ▶ **Q258 Upper advanced stop distance?** (incremental): Setup 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
- ▶ **Q259 Lower advanced stop distance?** (incremental): 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
- ▶ **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the TNC breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- ▶ **Q256 Retract dist. for chip breaking?** (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999



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 DEPTH
Q379=7.5	;STARTING POINT
Q253=750	;F PRE-POSITIONING
Q208=9999	;RETRACTION FEED RATE
Q395=0	;DEPTH REFERENCE

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

- ▶ **Q211 Dwell time at the depth?:** Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q379 Deepened starting point?** (incremental with respect to **Q203 SURFACE COORDINATE**, takes Q200 into account): Starting position of actual drilling. The TNC moves at **Q253 F PRE-POSITIONING** to the value **Q200 SET-UP CLEARANCE** above the deepened starting point. Input range 0 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?:** Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively **FMAX, FAUTO**
- ▶ **Q208 Feed rate for retraction?:** 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**
- ▶ **Q395 Diameter as reference (0/1)?:** 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**.

Fixed Cycles: Drilling

3.9 BORE MILLING (Cycle 208)

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!

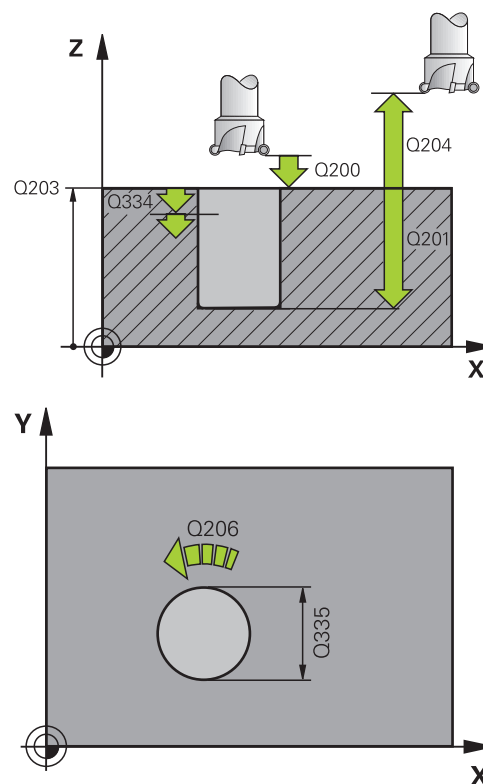
Enter in machine parameter displayDepthErr whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

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



- ▶ **Q200 Set-up clearance?** (incremental): Distance between underside of workpiece and the workpiece top surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during helical drilling. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q334 Feed per revolution of helix** (incremental): Depth of the tool plunge with each helix (=360°). Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q335 Nominal diameter?** (absolute): 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
- ▶ **Q342 Roughing diameter?** (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
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
+1 = Climb
-1 = Up-cut (if you enter 0, climb milling is performed)



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

Fixed Cycles: Drilling

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

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).

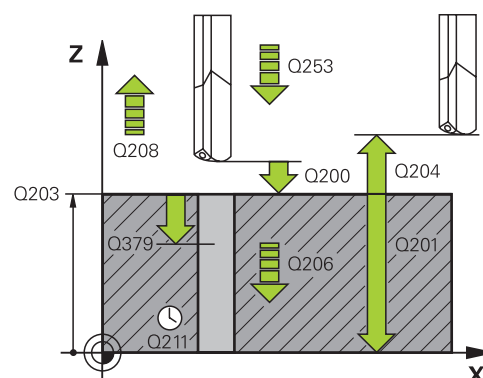
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and **Q203 SURFACE COORDINATE**. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between **Q203 SURFACE COORDINATE** and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO, FU**
- ▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Distance to machine datum Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q379 Deepened starting point?** (incremental with respect to **Q203 SURFACE COORDINATE**, takes Q200 into account): Starting position of actual drilling. The TNC moves at **Q253 F PRE-POSITIONING** to the value **Q200 SET-UP CLEARANCE** above the deepened starting point. Input range 0 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively **FMAX, FAUTO**
- ▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208=0**, the TNC retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input range 0 to 99999.999, alternatively **FMAX, FAUTO**
- ▶ **Q426 Rot. dir. of entry/exit (3/4/5)?**: 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
- ▶ **Q427 Spindle speed of entry/exit?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole Input range 0 to 99999



NC blocks

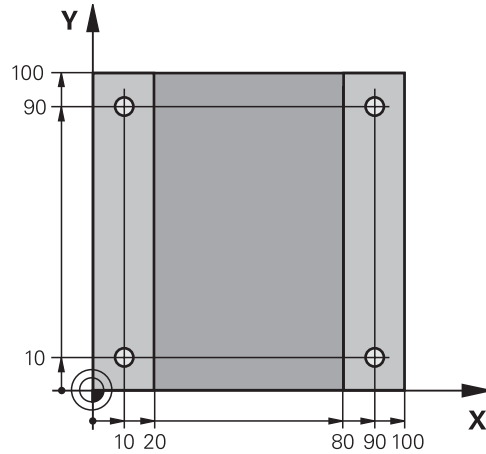
11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG	
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q211=0.25	;DWELL TIME AT DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q379=7.5	;STARTING POINT
Q253=750	;F PRE-POSITIONING
Q208=1000	;RETRACTION FEED RATE
Q426=3	;DIR. OF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

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

- ▶ **Q428 Spindle speed for drilling?:** Desired speed for drilling. Input range 0 to 99999
- ▶ **Q429 M function for coolant on?:** Miscellaneous function for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at **Q379 STARTING POINT**. Input range 0 to 999
- ▶ **Q430 M function for coolant off?:** Miscellaneous function for switching off the coolant. The TNC switches the coolant off if the tool is at **Q201 DEPTH**. Input range 0 to 999
- ▶ **Q435 Dwell depth? (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 **Q201 DEPTH**, input range 0 to 99999.9999.
- ▶ **Q401 Feed rate factor in %?:** Factor by which the TNC reduces the feed rate after **Q435 DWELL DEPTH** has been reached. Input range 0 to 100
- ▶ **Q202 Maximum plunging depth? (incremental):** Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range 0 to 99999.9999
- ▶ **Q212 Decrement? (incremental):** Value by which the TNC decreases **Q202 MAX. PLUNGING DEPTH** after each infeed. Input range 0 to 99999.9999
- ▶ **Q205 Minimum plunging depth? (incremental):** If you have entered **Q212 DECREMENT**, the TNC limits the plunging depth to the value for **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 DEPTH	
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	

Fixed Cycles: Drilling

3.11 Programming Examples

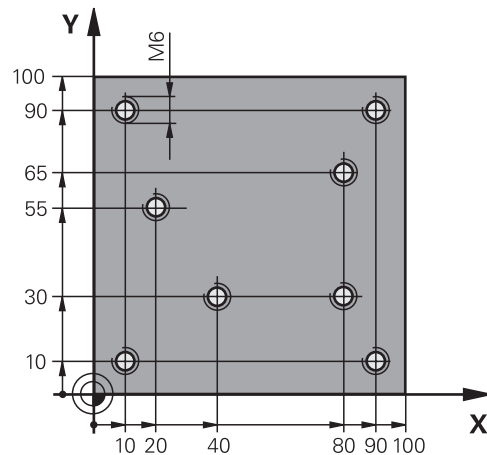
Example: Using drilling cycles in connection with PATTERN DEF

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	
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 DIA./DEPTH
Q201=-2	;DEPTH
Q344=-10	;DIAMETER
Q206=150	;FEED RATE FOR PLNGNG
Q211=0	;DWELL TIME AT DEPTH
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)

Programming Examples 3.11

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 DEPTH	
Q395=0 ;DEPTH REFERENCE	
12 CYCL CALL PAT F500 M13	Call the cycle in connection with the hole pattern
13 L Z+100 R0 FMAX	Retract the tool
14 TOOL CALL 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 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
17 CYCLE 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**



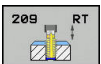

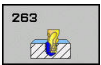

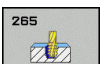

Fixed Cycles: Tapping / Thread Milling

4.1 Fundamentals

4.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	111
	207 TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	114
	209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	117
	262 THREAD MILLING Cycle for milling a thread in pre-drilled material	123
	263 THREAD MILLING/ COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	127
	264 THREAD DRILLING/MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool	131
	265 HELICAL THREAD DRILLING/ MILLING Cycle for milling the thread into solid material	135
	267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer	139

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.

Fixed Cycles: Tapping / Thread Milling

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

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!

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

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!

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

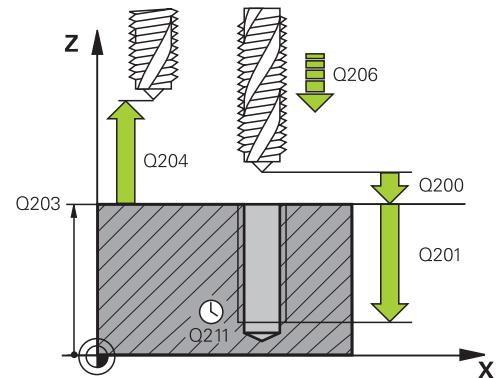
Cycle parameters



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

Guide value: 4x pitch.

- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during tapping. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q211 Dwell time at the depth?**: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (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 ;DEPTH OF THREAD

Q206=150 ;FEED RATE FOR
PLNGNG

Q211=0.25 ;DWELL TIME AT DEPTH

Q203=+25 ;SURFACE COORDINATE

Q204=50 ;2ND SET-UP
CLEARANCE

The feed rate is calculated as follows: $F = S \times p$

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.

Fixed Cycles: Tapping / Thread Milling

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

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 It then reverses the direction of spindle rotation again and the tool is retracted to the setup clearance. If you have entered a 2nd set-up clearance the TNC will move the tool with **FMAX** towards it.
- 4 The TNC stops the spindle turning at set-up clearance.

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

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.

If you program M3 (or M4) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the TOOL CALL block).

If you do not program M3 (or M4) before this cycle, the spindle stands still after the end of the cycle. Then you must restart the spindle with M3 (or M4) before the next operation.

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!

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

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!

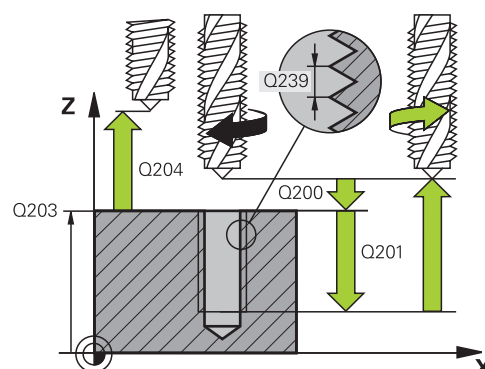
Fixed Cycles: Tapping / Thread Milling

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- ▶ **Q239 Pitch?**: 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
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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	;DEPTH OF THREAD
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. The TNC displays the **MANUAL TRAVERSE** soft key. After you pressed the **MANUAL TRAVERSE** soft key, you can retract the tool from 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 position it had assumed before the NC Stop key was pressed.



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.

Fixed Cycles: Tapping / Thread Milling

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

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.

With the parameter `CfgThreadSpindle>sourceOverride` you can set whether the potentiometer for the feed rate is in effect during thread cutting or not.

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.

If you program M3 (or M4) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the TOOL CALL block).

If you do not program M3 (or M4) before this cycle, the spindle stands still after the end of the cycle. Then you must restart the spindle with M3 (or M4) before the next operation.

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!

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

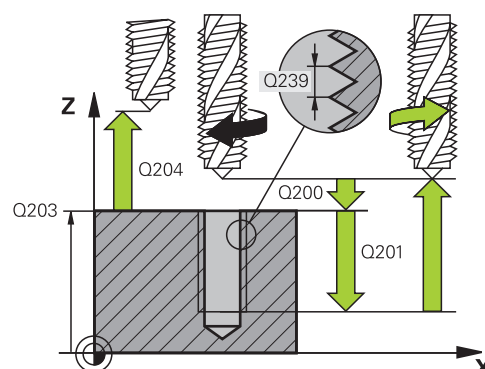
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- ▶ **Q239 Pitch?**: 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
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the TNC breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- ▶ **Q256 Retract dist. for chip breaking?**: 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
- ▶ **Q336 Angle for spindle orientation?** (absolute): Angle to 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
- ▶ **Q403 RPM factor for retraction?**: 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 OF THREAD
Q239=+1	;THREAD PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=+1	;DIST FOR CHIP BRKNG
Q336=50	;ANGLE OF SPINDLE
Q403=1.5	;RPM FACTOR

Fixed Cycles: Tapping / Thread Milling

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

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. The TNC displays the **MANUAL TRAVERSE** soft key. After you pressed the **MANUAL TRAVERSE** soft key, you can retract the tool from 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 position it had assumed before the NC Stop key was pressed.



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.

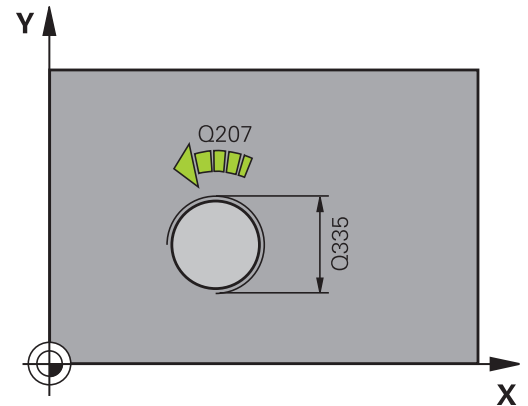
Procedure in case of a tool break

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.



Fixed Cycles: Tapping / Thread Milling

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

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.



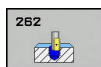
Danger of collision!

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

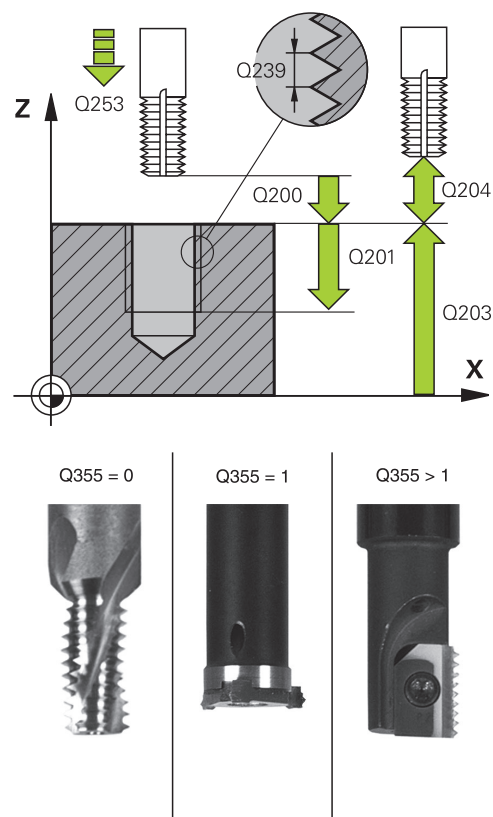
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



- ▶ **Q335 Nominal diameter?:** Thread inside diameter.
Input range 0 to 99999.9999
- ▶ **Q239 Pitch?:** 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
- ▶ **Q201 Depth of thread? (incremental):** Distance between workpiece surface and root of thread.
Input range -99999.9999 to 99999.9999
- ▶ **Q355 Number of threads per step?:** 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
- ▶ **Q253 Feed rate for pre-positioning?:** Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece.
Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** Type of milling operation with M3
 +1 = Climb
 -1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute):
Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999



NC blocks

25 CYCL DEF 262 THREAD MILLING

Q335=10 ;NOMINAL DIAMETER

Q239=+1.5 ;THREAD PITCH

Q201=-20 ;DEPTH OF THREAD

Q355=0 ;THREADS PER STEP

Fixed Cycles: Tapping / Thread Milling

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

- ▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- ▶ **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**

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
MILLNG**

Q512=0 ;FEED FOR APPROACH

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.

Fixed Cycles: Tapping / Thread Milling

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

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.



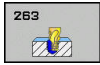
Danger of collision!

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

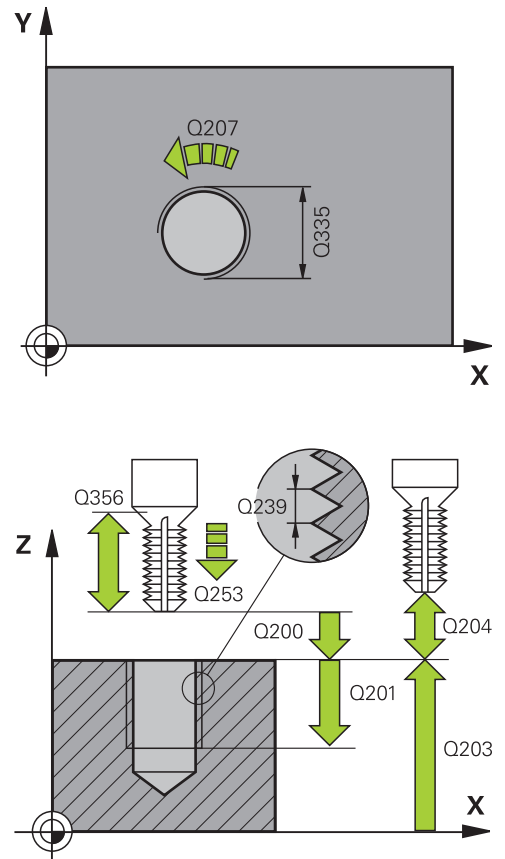
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters

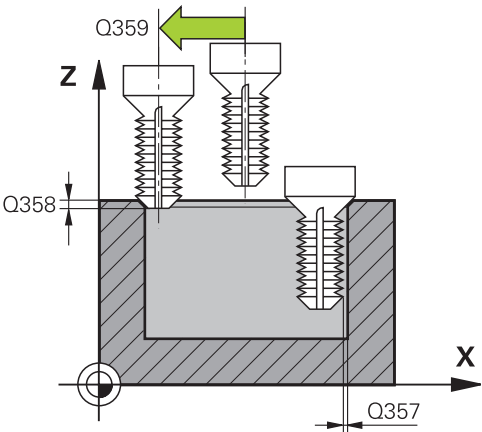


- ▶ **Q335 Nominal diameter?:** Thread inside diameter.
Input range 0 to 99999.9999
- ▶ **Q239 Pitch?:** Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 + = right-hand thread
 - = left-hand thread
 Input range -99.9999 to 99.9999
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread.
Input range -99999.9999 to 99999.9999
- ▶ **Q356 Countersinking depth?** (incremental): Distance between workpiece surface and tool tip.
Input range -99999.9999 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?:** Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece.
Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** Type of milling operation with M3
 +1 = Climb
 -1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q357 Safety clearance to the side?** (incremental): Distance between tool tooth and the wall. Input range 0 to 99999.9999
- ▶ **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- ▶ **Q359 Countersinking offset at front?** (incremental): Distance by which the TNC moves the tool center away from the center. Input range 0 to 99999.9999



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

- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- ▶ **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters you can decrease the approaching feed rate in order to reduce the danger of tool breakage. 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	;DEPTH OF THREAD
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 MILLNG
Q512=0	;FEED FOR APPROACH

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.

Fixed Cycles: Tapping / Thread Milling

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

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!

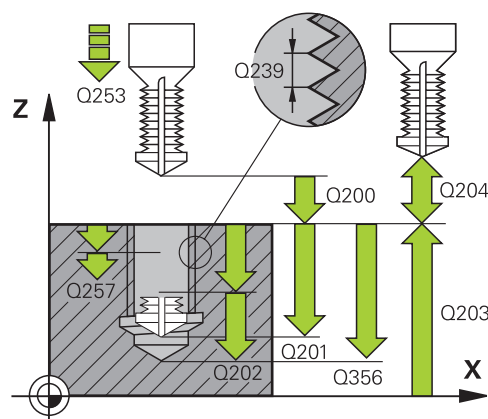
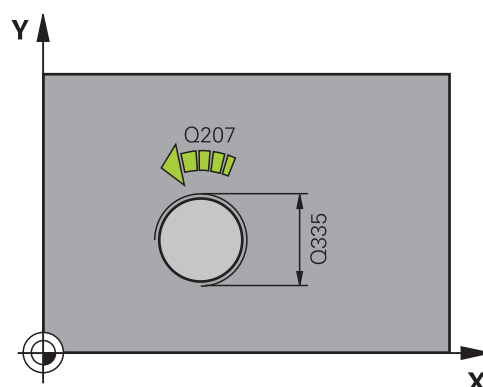
Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

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



- ▶ **Q335 Nominal diameter?:** Thread inside diameter.
Input range 0 to 99999.9999
- ▶ **Q239 Pitch?:** 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
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread.
Input range -99999.9999 to 99999.9999
- ▶ **Q356 Total hole depth?** (incremental): Distance between workpiece surface and hole bottom. Input range -99999.9999 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?:** Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece.
Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** Type of milling operation with M3
 +1 = Climb
 -1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q202 Maximum plunging depth?** (incremental): Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. 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
- ▶ **Q258 Upper advanced stop distance?** (incremental): Setup 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



NC blocks

25 CYCL DEF 264 THREAD DRILLNG/
MLLNG

Q335=10 ;NOMINAL DIAMETER

Q239=+1.5 ;THREAD PITCH

Q201=-16 ;DEPTH OF THREAD

Q356=-20 ;TOTAL HOLE DEPTH

Q253=750 ;F PRE-POSITIONING

Fixed Cycles: Tapping / Thread Milling

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

- ▶ **Q257 Infeed depth for chip breaking?**
(incremental): Plunging depth after which the TNC breaks the chip. No chip breaking if 0 is entered.
Input range 0 to 99999.9999
- ▶ **Q256 Retract dist. for chip breaking?**
(incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999
- ▶ **Q358 Sinking depth at front?** (incremental):
Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- ▶ **Q359 Countersinking offset at front?**
(incremental): Distance by which the TNC moves the tool center away from the center. Input range 0 to 99999.9999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute):
Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.999 alternatively **FAUTO, FU**
- ▶ **Q207 Feed rate for milling?:** Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- ▶ **Q512 Feed rate for approaching?:** Traversing speed of the tool in mm/min while approaching. For smaller thread diameters you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**

Q351=+1	;CLIMB OR UP-CUT
Q202=5	;PLUNGING DEPTH
Q258=0.2	;UPPER 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 MILLNG
Q512=0	;FEED FOR APPROACH

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.

Fixed Cycles: Tapping / Thread Milling

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

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!

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

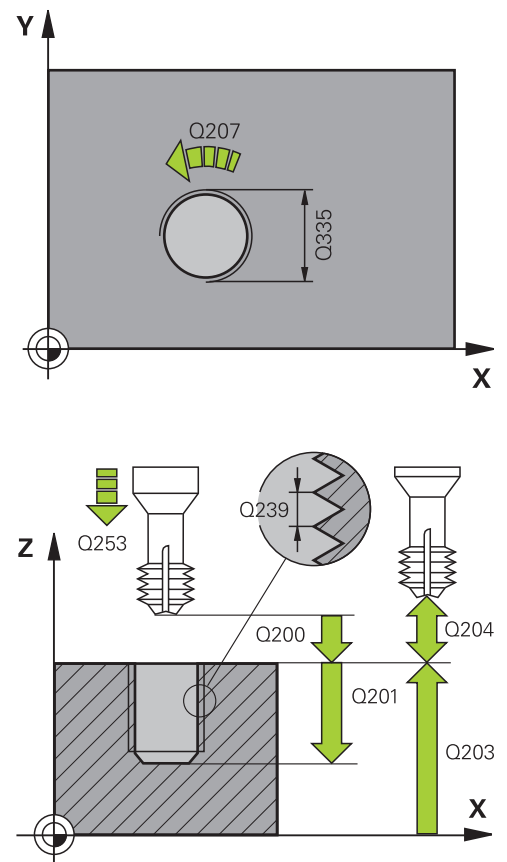
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



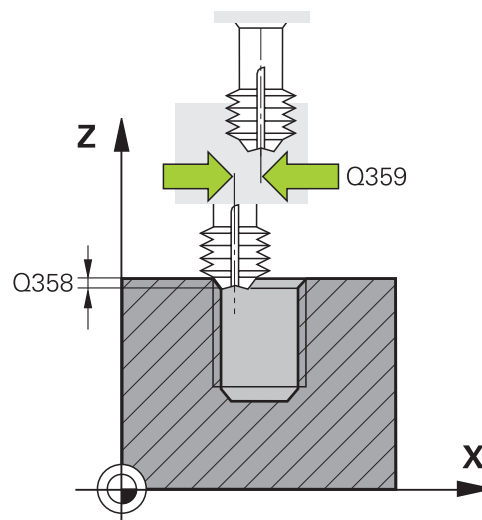
- ▶ **Q335 Nominal diameter?:** Thread inside diameter. Input range 0 to 99999.9999
- ▶ **Q239 Pitch?:** Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 + = right-hand thread
 - = left-hand thread
 Input range -99.9999 to 99.9999
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?:** Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- ▶ **Q359 Countersinking offset at front?** (incremental): Distance by which the TNC moves the tool center away from the center. Input range 0 to 99999.9999
- ▶ **Q360 Countersink (before/after:0/1)?** : Running the chamfer
 0 = before thread milling
 1 = after thread milling
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999



Fixed Cycles: Tapping / Thread Milling

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

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during 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	;DEPTH OF THREAD
Q253=750	;F PRE-POSITIONING
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q360=0	;COUNTERSINK PROCESS
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLNG

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.

Fixed Cycles: Tapping / Thread Milling

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

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!

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

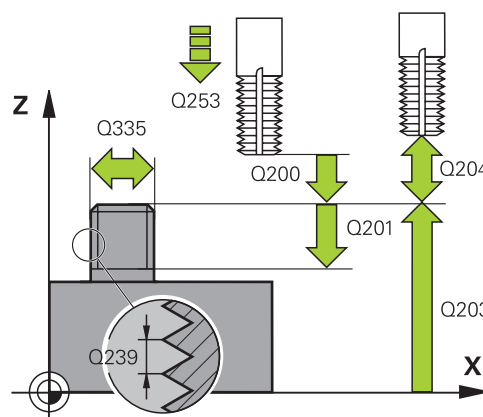
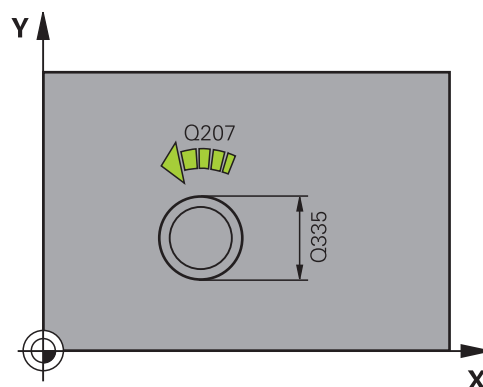
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

Cycle parameters



- ▶ **Q335 Nominal diameter?:** Thread inside diameter. Input range 0 to 99999.9999
- ▶ **Q239 Pitch?:** 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
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- ▶ **Q355 Number of threads per step?:** 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
- ▶ **Q253 Feed rate for pre-positioning?:** Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** Type of milling operation with M3
 +1 = Climb
 -1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- ▶ **Q359 Countersinking offset at front?** (incremental): Distance by which the TNC moves the tool center away from the center. Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



Q355 = 0



Q355 = 1



Q355 > 1



NC blocks

25 CYCL DEF 267 OUTSIDE THREAD
MLLNG

Q335=10 ;NOMINAL DIAMETER

Q239=+1.5 ;THREAD PITCH

Q201=-20 ;DEPTH OF THREAD

Fixed Cycles: Tapping / Thread Milling

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

- ▶ **Q254 Feed rate for counterboring?:** Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- ▶ **Q207 Feed rate for milling?:** Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- ▶ **Q512 Feed rate for approaching?:** Traversing speed of the tool in mm/min while approaching. For smaller thread diameters you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**

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 MILLNG
Q512=0	;FEED FOR APPROACH

4.11 Programming Examples

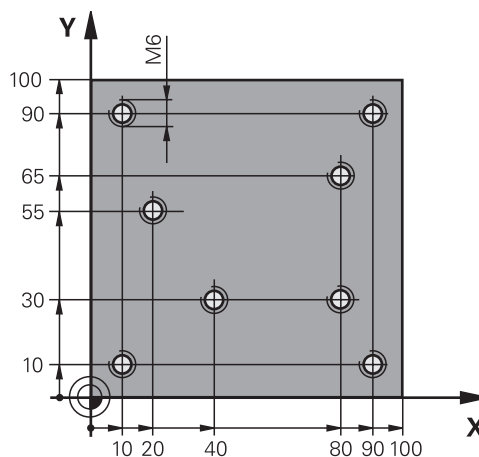
Example: Thread milling

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 Z+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

Fixed Cycles: Tapping / Thread Milling

4.11 Programming Examples

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. PNTMM
NRXYZ
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

5.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	147
	252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	152
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal plunging	157
	254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal plunging	162
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	167
	257 CIRCULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	171
	233 FACE MILLING Machining the face with up to 3 limits	180

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 path overlap (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 pecking 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 have been defined, the TNC plunges and then approaches the contour. The approach movement occurs on a radius in order to ensure that a gentle approach is possible. The TNC first finishes the pocket walls, in multiple infeeds if so specified.
- 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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!**

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

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!

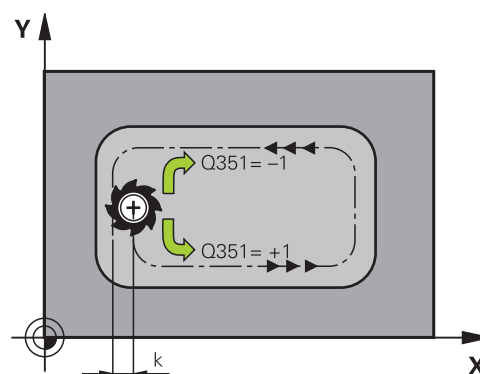
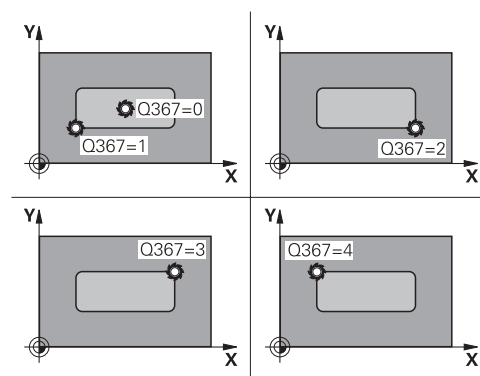
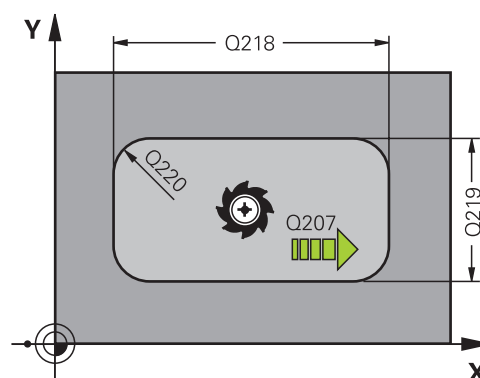
If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse!

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

Cycle parameters



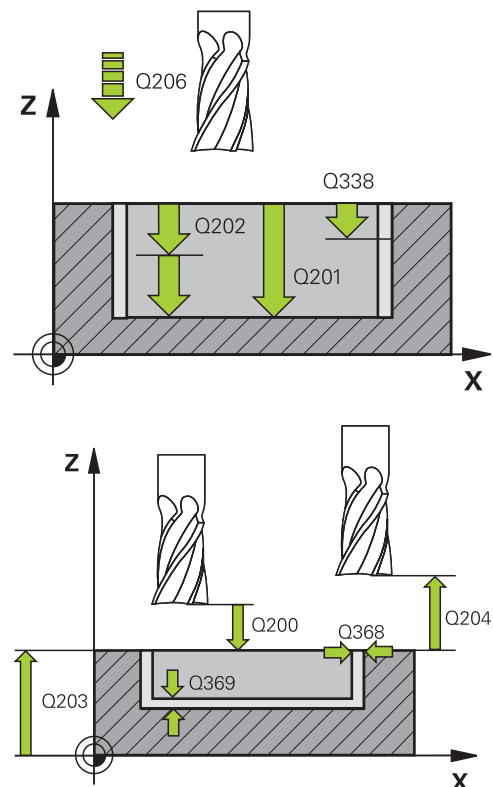
- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
 Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- ▶ **Q218 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane Input range 0 to 99999.9999
- ▶ **Q219 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane Input range 0 to 99999.9999
- ▶ **Q220 Corner radius?**: 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
- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q224 Angle of rotation?** (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
- ▶ **Q367 Position of pocket (0/1/2/3/4)?**: 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 corner top
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket Input range -99999.9999 to 99999.9999



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut $Q338=0$: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q370 Path overlap factor?**: $Q370 \times \text{tool radius} = \text{stepover factor } k$. Input range: 0.1 to 1.414; alternatively **PREDEF**
- ▶ **Q366 Plunging strategy (0/1/2)?**: 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



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 MILLNG
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	;INFEEED FOR FINISHING

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

- ▶ **Q385 Finishing feed rate?:** Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q439 Feed rate reference (0-3)?:** Specify what the programmed feed rate refers to:
 - 0:** Feed rate with respect to the tool center point path
 - 1:** Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center point path
 - 2:** Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise it refers to the tool path center
 - 3:** Feed rate always refers to the cutting edge

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=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

5.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)**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 path overlap (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.

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

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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!**

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

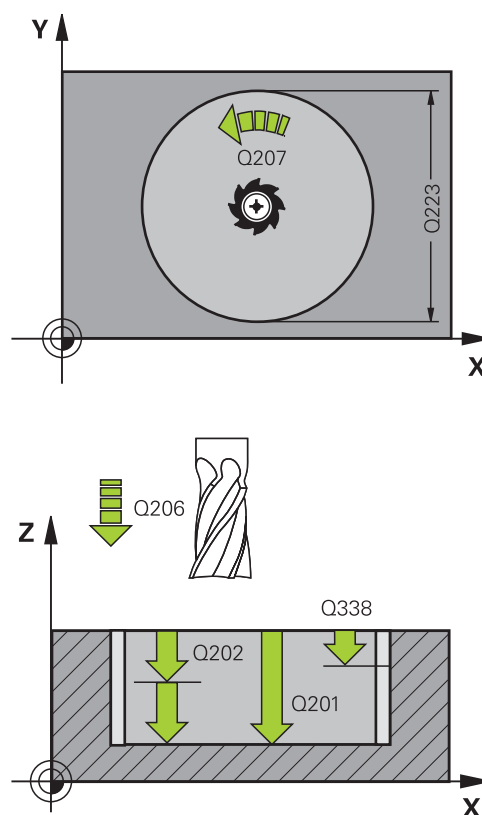
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse!

Cycle parameters



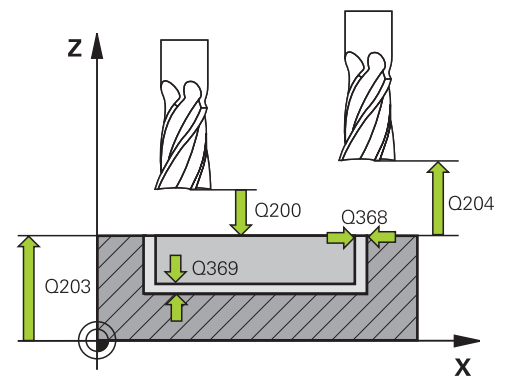
- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
 Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- ▶ **Q223 Circle diameter?**: Diameter of the finished pocket. Input range 0 to 99999.9999
- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q370 Path overlap factor?:** $Q370 \times \text{tool radius} = \text{stepover factor } k$. Input range: 0.1 to 1.9999; alternatively **PREDEF**
- ▶ **Q366 Plunging strategy (0/1)?:** 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**
- ▶ **Q385 Finishing feed rate?:** Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q439 Feed rate reference (0-3)?:** Specify what the programmed feed rate refers to:
 - 0:** Feed rate with respect to the tool center point path
 - 1:** Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center point path
 - 2:** Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise it refers to the tool path center
 - 3:** Feed rate always refers to the 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 MILLNG
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	;INFEEED 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 (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 in the left slot arc.
- 6 Then the TNC finishes the floor of the slot from the inside out.

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

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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!**

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

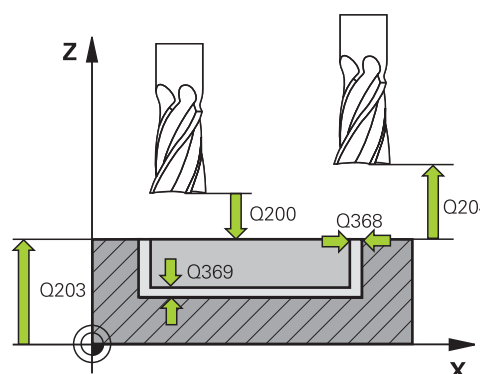
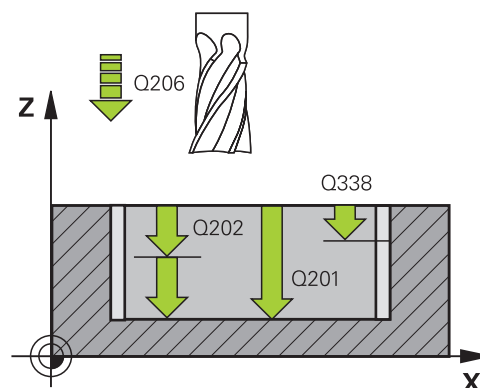
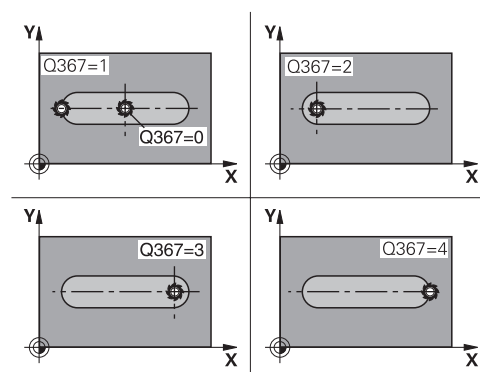
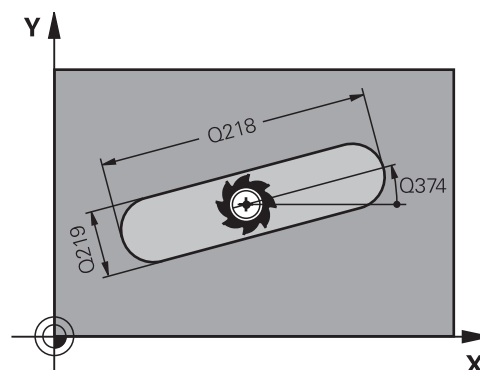
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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



- ▶ **Q215 Machining operation (0/1/2)?**: 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
- ▶ **Q218 Length of slot?** (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range 0 to 99999.9999
- ▶ **Q219 Width of slot?** (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
- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q374 Angle of rotation?** (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
- ▶ **Q367 Position of slot (0/1/2/3/4)?**: 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
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor Input range 0 to 99999.9999



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q366 Plunging strategy (0/1/2)?:** 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**
- ▶ **Q385 Finishing feed rate?:** Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**

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 MILLNG
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	;INFEEED 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

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

- **Q439 Feed rate reference (0-3)?:** Specify what the programmed feed rate refers to:

- 0:** Feed rate with respect to the tool center point path
- 1:** Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center point path
- 2:** Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise it refers to the tool path center
- 3:** Feed rate always refers to the cutting edge

Q439=0 ;FEED RATE REFERENCE

9 L X+50 Y+50 R0 FMAX M3 M99

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!**

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

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!

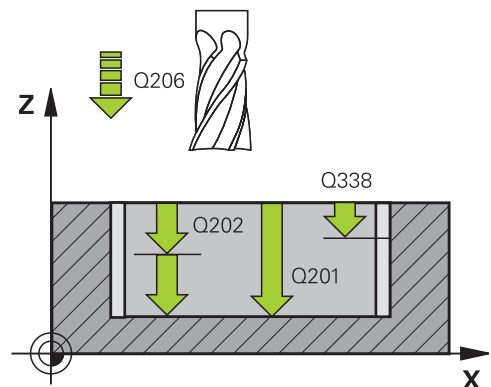
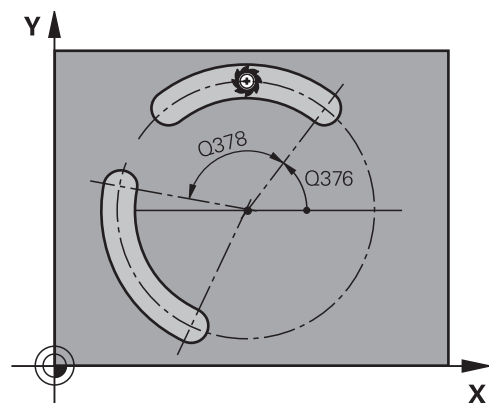
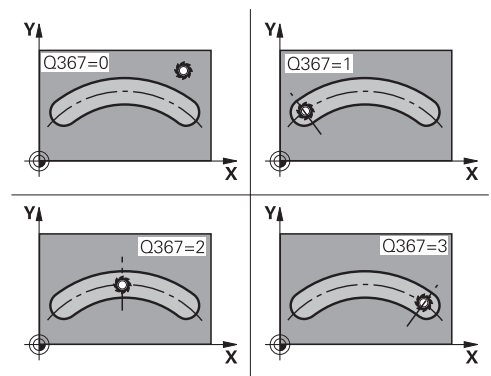
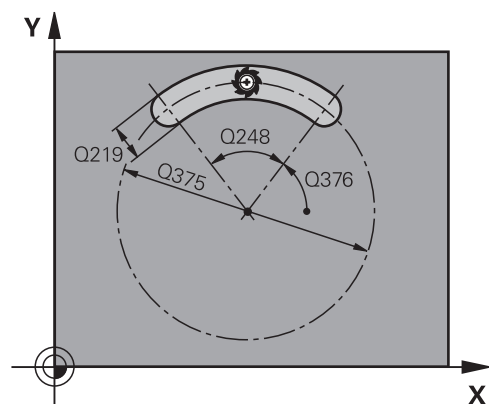
Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

Cycle parameters

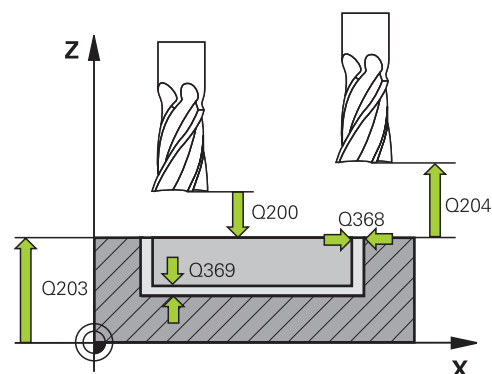


- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
 Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- ▶ **Q219 Width of slot?** (value parallel to the secondary axis 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
- ▶ **Q368 Finishing allowance for side?** (incremental):
 Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q375 Pitch circle diameter?**: Enter the diameter of the pitch circle. Input range 0 to 99999.9999
- ▶ **Q367 Ref. for slot pos. (0/1/2/3)?**: 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.
- ▶ **Q216 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane.
Only effective if Q367 = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q217 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane.
Only effective if Q367 = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q376 Starting angle?** (absolute): Enter the polar angle of the starting point. Input range -360.000 to 360.000
- ▶ **Q248 Angular length?** (incremental): Enter the angular length of the slot. Input range 0 to 360.000
- ▶ **Q378 Intermediate stepping angle?** (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
- ▶ **Q377 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range 1 to 99999



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

- ▶ **Q207 Feed rate for milling?** Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?** Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 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 DIAMETR
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 MILLNG
Q351=+1	;CLIMB OR UP-CUT

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.
Input range 0 to 99999.9999
- ▶ **Q366 Plunging strategy (0/1/2)?**: 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
- ▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
0: Feed rate with respect to the tool center point path
1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center point path
2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise it refers to the tool path center
3: Feed rate always refers to the cutting edge

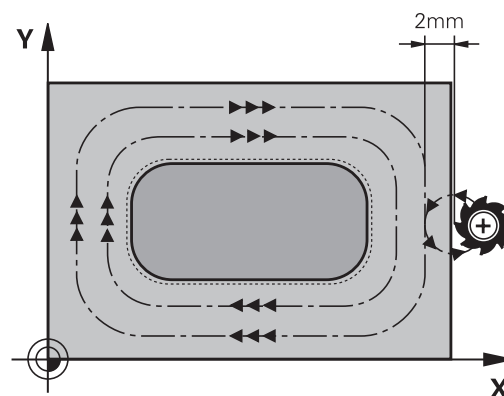
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 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 setting (**Q437=0**) lies 2 mm to the right next to the stud blank
- 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, on the other hand, you did not set the starting point on a side, but rather 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 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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!**

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

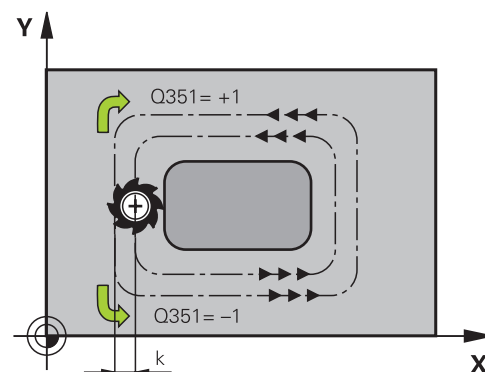
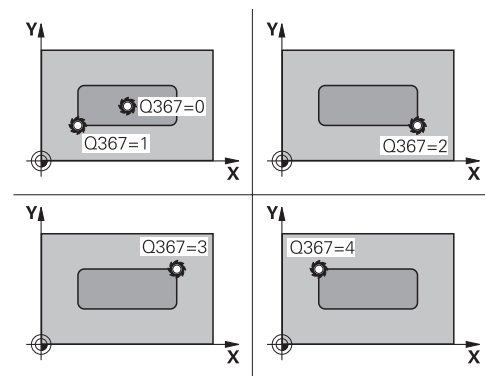
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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. The end position of the tool after the cycle differs from the starting position!

256

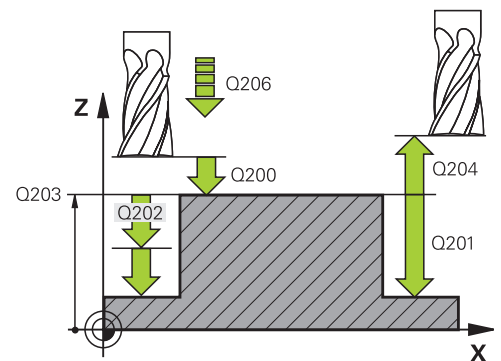
-



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999; alternatively **FMAX, FAUTO, FU, FZ**
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q370 Path overlap factor?:** Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.9999; alternatively **PREDEF**
- ▶ **Q437 Starting position (0...4)?:** 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 should be appear on the stud surface during approach with the setting Q437=0, then choose 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 MILLNG
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 path overlap 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.

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!

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

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!

The TNC performs an approach motion in this cycle! Depending on the starting angle Q376, the following amount of space must be left next to the stud: At least tool diameter + 2 mm. Danger of collision!

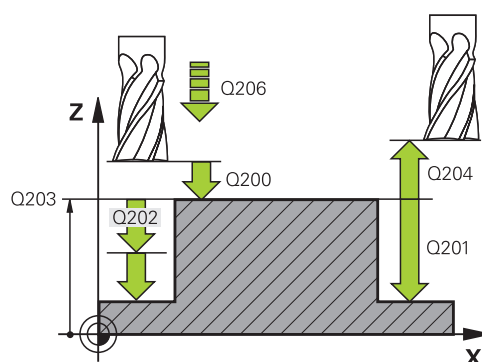
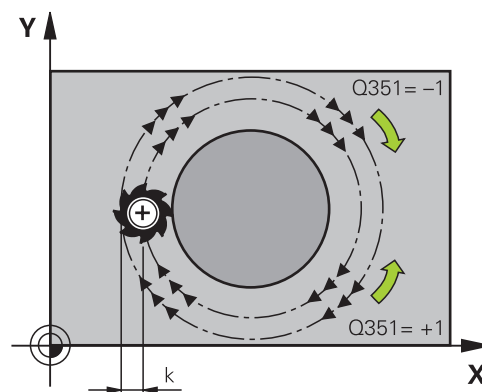
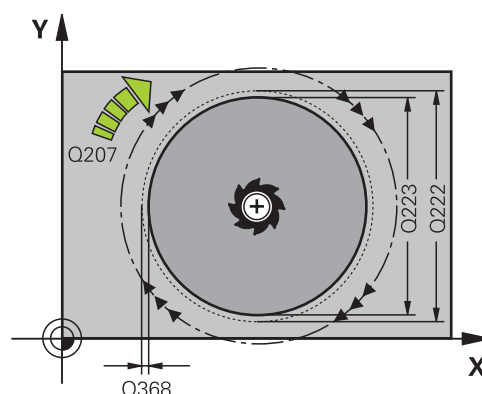
At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position!

Enter a starting angle between 0° and 360° in parameter Q376 in order to determine the exact starting position. If you use the default value -1 the TNC automatically calculates the most favorable starting position. These may vary

Cycle parameters



- ▶ **Q223 Finished part diameter?:** Diameter of the completely machined stud. Input range 0 to 99999.9999
- ▶ **Q222 Workpiece blank diameter?:** 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
- ▶ **Q368 Finishing allowance for side? (incremental):** Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q207 Feed rate for milling?:** Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1:** 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 performed)
- ▶ **Q201 Depth? (incremental):** Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth? (incremental):** Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999; alternatively **FMAX, FAUTO, FU, FZ**
- ▶ **Q200 Set-up clearance? (incremental):** Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q203 Workpiece surface coordinate?** (absolute):
Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q370 Path overlap factor?:** Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414; alternatively **PREDEF**
- ▶ **Q376 Starting angle?:** Polar angle relative to the stud center from which the tool approaches the stud. Input range 0 to 359°

NC blocks

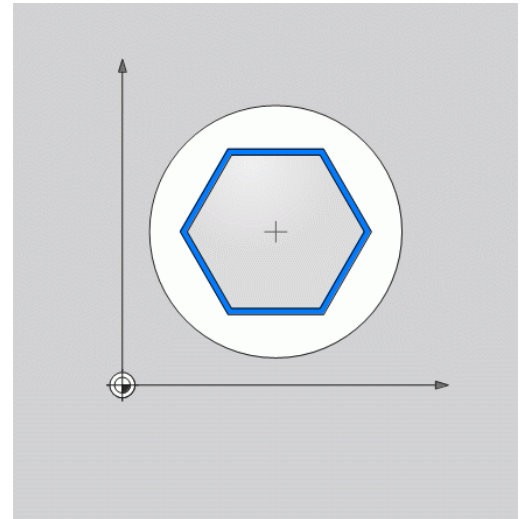
8 CYCL DEF 257 CIRCULAR STUD	
Q223=60	;FINISHED PART DIA.
Q222=60	;WORKPIECE BLANK DIA.
Q368=0.2	;ALLOWANCE FOR SIDE
Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;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 POLYGON STUD (Cycle 258, DIN/ISO: G258)

Cycle run

With the cycle **POLYGON STUD** you can create an even polygon by machining the contour outside. The milling operation is carried out on a spiral path, based on the diameter of the workpiece blank.

- 1 If, at the beginning of machining, the work piece is positioned below the 2nd set-up clearance, the TNC will retract the tool back to the 2nd setup clearance.
- 2 Starting from the center of the stud the TNC moves the tool to the starting point of the stud machining. The starting point depends, among others, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter Q224.
- 3 The tool moves at rapid traverse **FMAX** to the setup clearance Q200 and from there with the feed rate for plunging to the first plunging depth.
- 4 Then the TNC creates the polygon stud in a spiral-shaped pass, taking into account the path overlap
- 5 The TNC moves the tool on a tangential path from the outside to the inside
- 6 The tool will be lifted in the direction of the spindle axis to the 2nd setup clearance in one rapid movement
- 7 If several plunging depths are required, the TNC will position the tool back to the starting point of the stud milling process.
- 8 This process is repeated until the programmed stud depth is reached.
- 9 At the end of the cycle first a departing motion is performed. Then the TNC will move the tool on the tool axis to the 2nd setup clearance.



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.8 POLYGON STUD (Cycle 258, DIN/ISO: G258)

Please note while programming:



Before the start of the cycle you will have to preposition the tool on the machining plane. In order to do so, move the tool with radius compensation **R0** to the center of the stud.

The TNC automatically pre-positions the tool in the tool axis. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

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!

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

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!

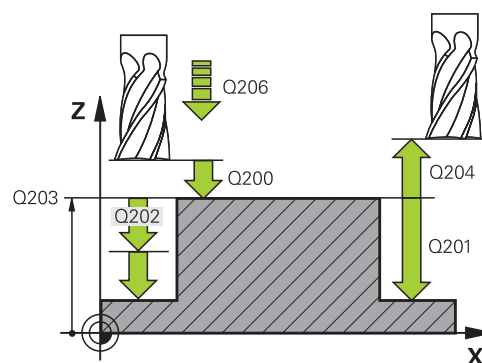
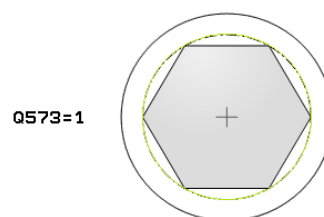
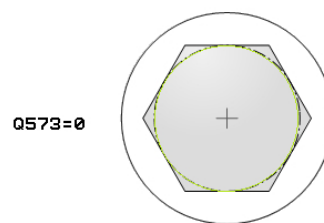
The TNC performs an approaching motion in this cycle! Depending on the rotary position Q224, the following amount of space must be left next to the stud: At least tool diameter + 2mm. Danger of collision!

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. The end position of the tool after the cycle differs from the starting position!

Cycle parameters



- ▶ **Q573 Inscr.circle/circumcircle (0/1)?**: Definition of whether the dimensioning shall reference to the inscribed circle or to the perimeter:
0= dimensioning refers to the inscribed circle
1= dimensioning refers to the perimeter
- ▶ **Q571 Reference circle diameter?**: Definition of the diameter of the reference circuit. Specify in parameter Q573 whether the diameter references to the inscribed circle or the perimeter. Input range: 0 to 99999.9999
- ▶ **Q222 Workpiece blank diameter?**: Definition of the diameter of the workpiece blank. The workpiece blank diameter must be greater than the reference circle diameter. The TNC performs multiple stepovers if the difference between the workpiece blank diameter and reference circle 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
- ▶ **Q572 Number of corners?**: Enter the number of corners of the polygon. The TNC will always equally divide the corners on the stud. Input range 3 to 30
- ▶ **Q224 Angle of rotation?**: Specify which angle is used to machine the first corner of the polygon. Input range: -360° to +360°



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.8 POLYGON STUD (Cycle 258, DIN/ISO: G258)

- ▶ **Q220 Radius / Chamfer (+/-)?**: Enter the value for the input form radius or chamfer. If you enter a positive value between 0 and +99999.9999, the TNC rounds every corner of the polygon stud. The radius refers to the value you entered. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.
- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999; alternatively **FMAX, FAUTO, FU, FZ**
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

NC blocks

8 CYCL DEF 258 POLYGON STUD	
Q573=1	;REFERENCE CIRCLE
Q571=50	;REF-CIRCLE DIAMETER
Q222=120	;WORKPIECE BLANK DIA.
Q572=10	;NUMBER OF CORNERS
Q224=40	;ANGLE OF ROTATION
Q220=2	;RADIUS / CHAMFER
Q368=0	;ALLOWANCE FOR SIDE
Q207=3000	;FEED RATE FOR MILLNG
Q351=1	;CLIMB OR UP-CUT
Q201=-18	;DEPTH
Q202=10	;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
Q215=0	;MACHINING OPERATION
Q369=0	;ALLOWANCE FOR FLOOR
Q338=0	;INFEEED FOR FINISHING
Q385=500	;FINISHING FEED RATE
9 L X+50 Y+50 R0 FMAX M3 M99	

- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q370 Path overlap factor?:** $Q370 \times \text{tool radius} = \text{stepover factor } k$. Input range: 0.1 to 1.414;
alternatively **PREDEF**
- ▶ **Q215 Machining operation (0/1/2)?:** 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
- ▶ **Q369 Finishing allowance for floor?** (incremental):
Finishing allowance for the floor Input range 0 to 99999.9999
- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut $Q338=0$: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Q385 Finishing feed rate?:** Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**

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

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

Cycle run

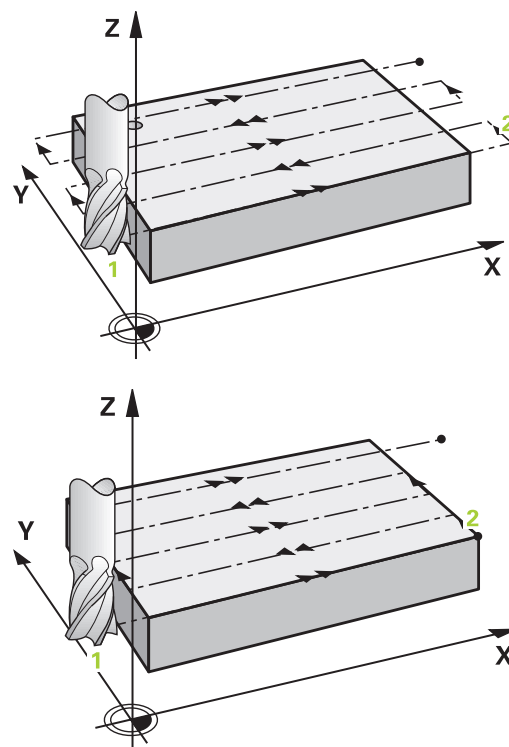
Cycle 233 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Additionally, you can also define side walls in the cycle which are taken into account during the machining of the level surface. The cycle offers you various machining strategies:

- **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 tool 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.

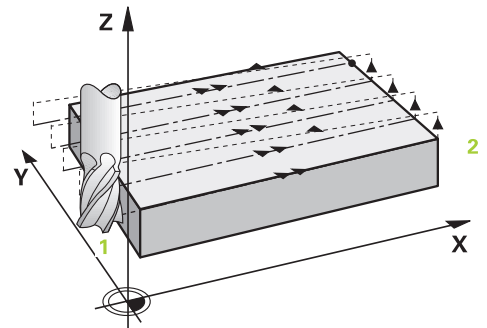
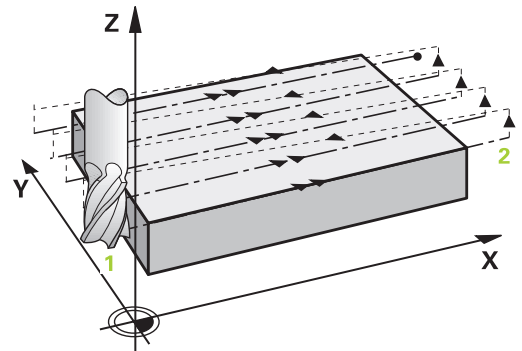


5.9 FACE MILLING (Cycle 233, DIN/ISO: G233)

Strategies Q389=2 and Q389=3

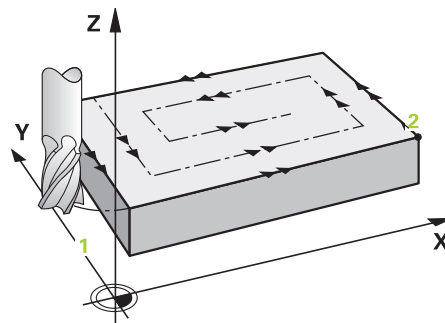
The strategies Q389=2 and Q389=3 differ in the overtravel during face milling. If Q389=2, the end point lies outside of the surface. If Q389=3, it lies at the edge of the surface. The TNC calculates the end point **2** from the side length and the safety clearance to the side. If the strategy Q389=2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

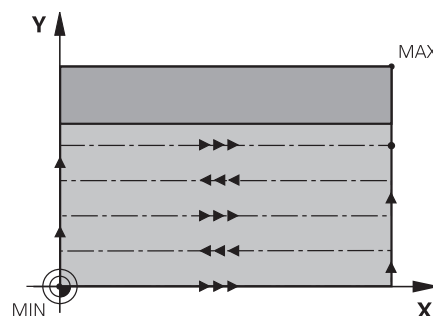


Strategy Q389=4

- 4 The tool subsequently approaches the starting point of the milling path at the programmed **Feed rate for milling** on a tangential arc.
- 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 tool 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 limiters enable you to limit the machining of the level surface, for example, to account for side walls or shoulders 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. For roughing the TNC includes the oversize of the side - for finishing the oversize helps to preposition the tool.



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

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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

If **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS** are entered as equal values, the TNC does not run the cycle (depth = 0 has been programmed).

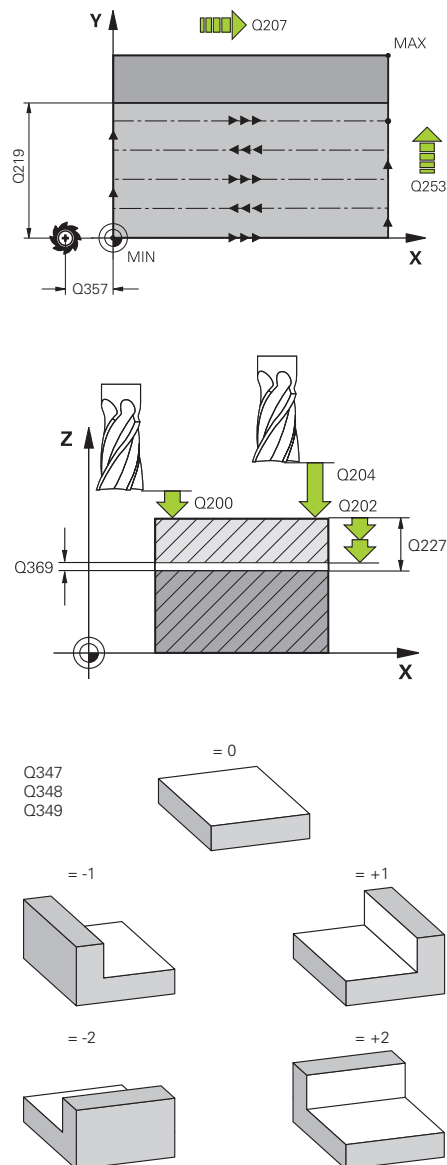
**Danger of collision!**

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

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

233

- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
 - 0**: Roughing and finishing
 - 1**: Only roughing
 - 2**: Only finishingSide finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is deined
- ▶ **Q389 Machining strategy (0-4)?**: 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
 - 3**: Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined
 - 4**: Helical machining, uniform infeed from the outside toward the inside
- ▶ **Q350 Milling direction?**: Axis in the machining plane that defines the machining direction:
 - 1**: Reference axis = machining direction
 - 2**: Minor axis = machining direction
- ▶ **Q218 First side length?** (incremental): Length of the surface to be machined in the reference axis of the working plane, referenced to the starting point in the 1st axis. Input range -99999.9999 to 99999.9999
- ▶ **Q219 Second side length?** (incremental): 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 **STARTNG PNT 2ND AXIS**. Input range -99999.9999 to 99999.9999
- ▶ **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999



Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

- ▶ **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis on which the surface is to be face-milled. Input range -99999.9999 to 99999.9999
- ▶ **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range 0 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Q370 Path overlap factor?**: 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.
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q253 Feed rate for pre-positioning?**: 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**
- ▶ **Q357 Safety clearance to the side?** (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
- ▶ **Q200 Set-up clearance?** (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	;FIRST 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 MILLNG
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO 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=2	;CORNER RADIUS
Q368=0	;ALLOWANCE FOR SIDE
Q338=0	;INFEEED FOR FINISHING
9 L X+0 Y+0 R0 FMAX M3 M99	

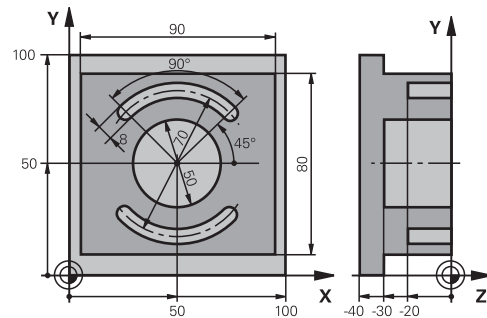
- ▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.
Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q347 1st limit?**: Select the side of the workpiece where the plan surface is bordered by a side wall (not possible with 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 with helical machining):
Input **0**: No limiting
Input **-1**: Limiting in negative principal axis
Input **+1**: Limiting in positive principal axis
Input **-2**: Limiting in negative secondary axis
Input **+2**: Limiting in positive secondary axis
- ▶ **Q348 2nd limit?**: See parameter for 1st limit Q347
- ▶ **Q349 3rd limit?**: See parameter for 1st limit Q347
- ▶ **Q220 Corner radius?**: Radius of a corner at limits (Q347 to Q349). Input range 0 to 99999.9999
- ▶ **Q368 Finishing allowance for side?** (incremental):
Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut Q338=0: Finishing in one infeed. Input range 0 to 99999.9999

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.10 Programming Examples

5.10 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 MILLNG	
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 MILLNG	

Programming Examples 5.10

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	;INFEEED 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	
Q439=0	;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99		Call CIRCULAR POCKET MILLING cycle
9 L Z+250 R0 FMAX M6		Tool change
10 TOOL 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 DIAMETR	
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 MILLNG	
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	;INFEEED 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	
12 CYCL CALL FMAX M3		Call SLOT cycle

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.10 Programming Examples

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

6.1


Fundamentals

Overview

The TNC provides two cycles for machining point patterns directly:

Soft key	Cycle	Page
	220 POLAR PATTERN	193
	221 CARTESIAN PATTERN	196

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 73) to develop point tables.

More regular point patterns are available with the **PATTERN DEF** function (see "PATTERN DEF pattern definition", page 66).

- 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:
 - 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 a 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:



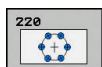
Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

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 the 2nd set-up clearance that were defined in Cycle 220 will be effective.

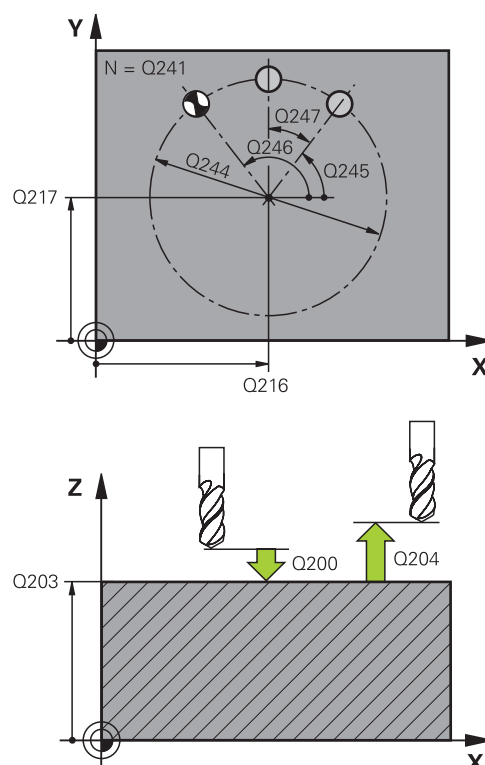
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

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

Cycle parameters



- ▶ **Q216 Center in 1st axis?** (absolute): Pitch circle center in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range 0 to 99999.9999
- ▶ **Q245 Starting angle?** (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
- ▶ **Q246 Stopping angle?** (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 complete 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
- ▶ **Q247 Intermediate stepping angle?** (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
- ▶ **Q241 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range 1 to 99999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q203 Workpiece surface coordinate?** (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
DIAMETR

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

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

- ▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.
Input range 0 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q365 Type of traverse? Line=0/arc=1**: 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

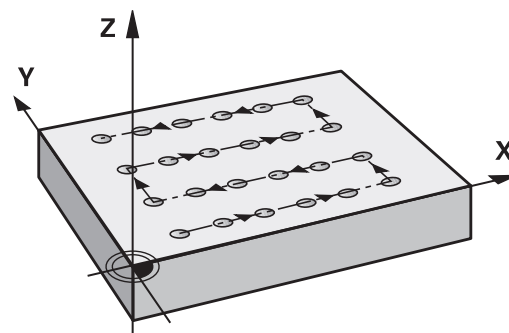
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE
Q365=0	;TYPE OF TRAVERSE

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

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:
 - Move to the 2nd 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.

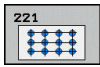
If you combine Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface, the 2nd set-up clearance, and the rotational position that were defined in Cycle 221 will be effective.

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

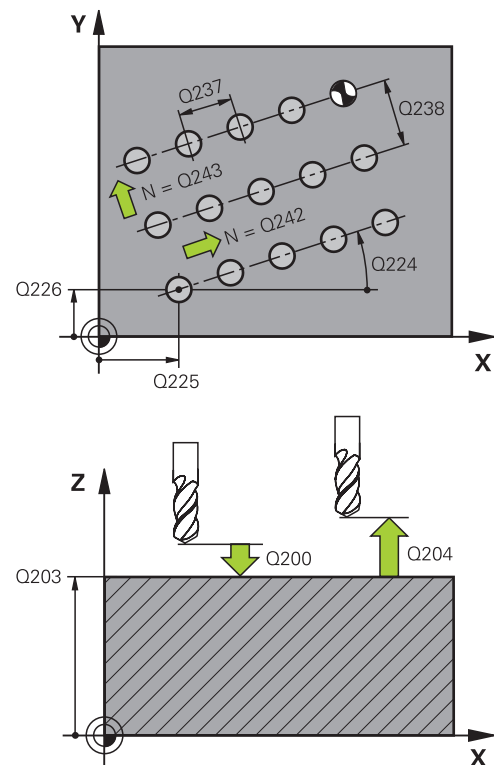
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

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

Cycle parameters



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

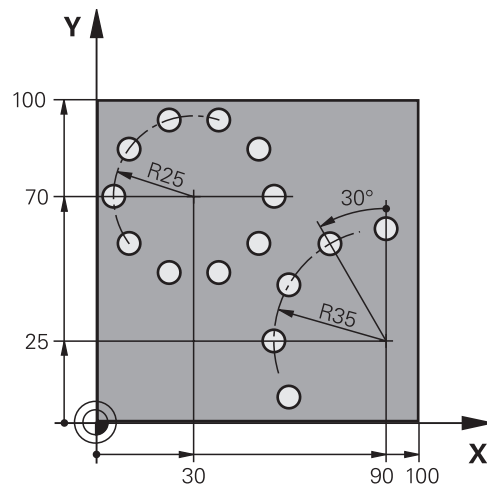
Q301=1 ;MOVE TO CLEARANCE

6 Fixed Cycles: Pattern Definitions

6.4 Programming Examples

6.4 Programming Examples

Example: Polar hole patterns



0 BEGIN PGM HOLEPAT 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	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 DEPTH	
Q395=0 ;DEPTH REFERENCE	
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 DIAMETR	
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	

Programming Examples 6.4

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 DIAMETR	
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 HOLEPAT MM		

7

**Fixed Cycles:
Contour Pocket**

7.1 SL Cycles

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 GEOMETRY ...
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
...

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before each cycle. You must move 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.

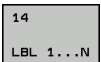





The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

60 LBL 0


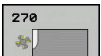
...

99 END PGM SL2 MM

Overview

Soft key	Cycle	Page
	14 CONTOUR GEOMETRY (compulsory)	204
	20 CONTOUR DATA (essential)	209
	21 PILOT DRILLING (optional)	211
	22 ROUGH-OUT (compulsory)	213
	23 FLOOR FINISHING (optional)	217
	24 SIDE FINISHING (optional)	219

Enhanced cycles:

Soft key	Cycle	Page
	25 CONTOUR TRAIN	222
	270 CONTOUR TRAIN DATA	224

Fixed Cycles: Contour Pocket

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

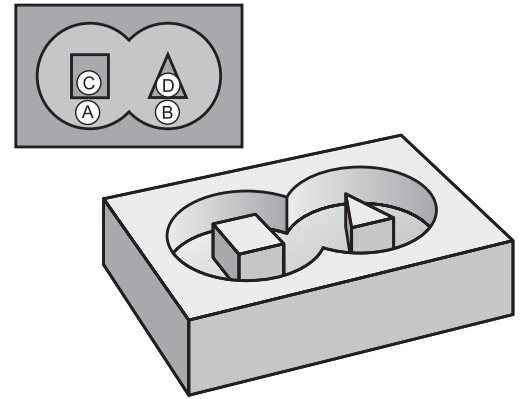
7.2 CONTOUR (Cycle 14, DIN/ISO: G37)

Please note while programming:

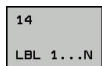
All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY.



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

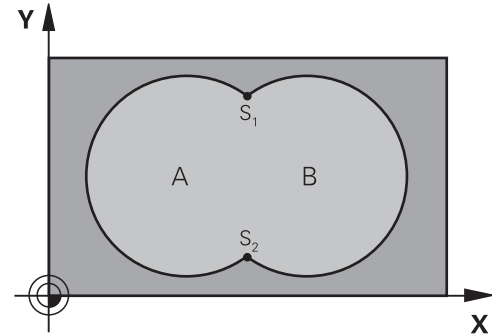


- **Label numbers for the contour:** Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the **END** key. Entry of up to 12 subprogram numbers 1 to 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  
    LABEL1/2/3/4
```

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 need not 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
```

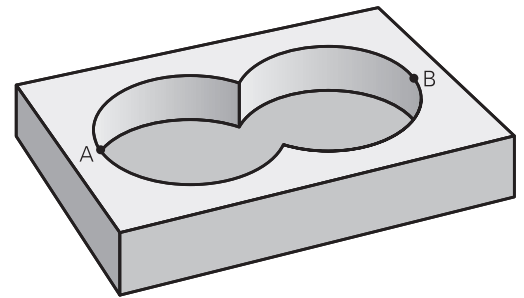
Fixed Cycles: Contour Pocket

7.3 Superimposed contours

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

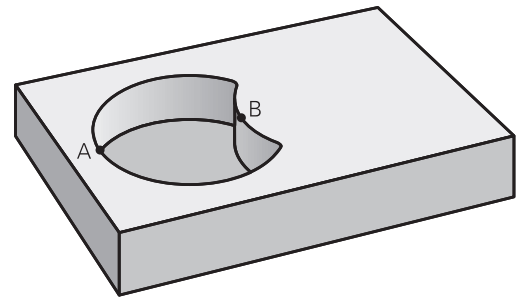
Surface B:

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

7

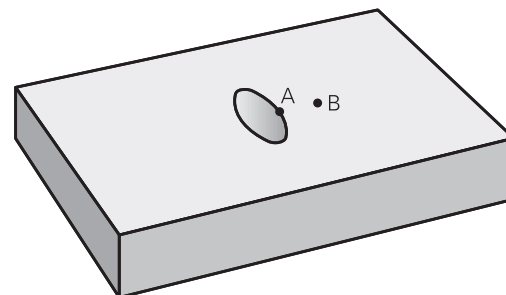
Fixed Cycles: Contour Pocket

7.3 Superimposed contours

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.

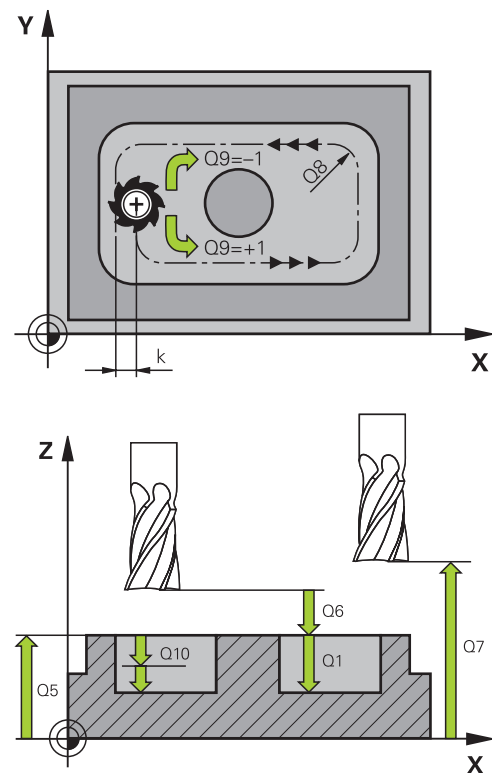
Fixed Cycles: Contour Pocket

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

Cycle parameters

28
CONTOUR
DATA

- ▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- ▶ **Q2 Path overlap factor?** $Q2 \times \text{tool radius} = \text{stepover factor } k$. Input range: -0.0001 to 1.9999
- ▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range -99999.9999 to 99999.9999
- ▶ **Q4 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range -99999.9999 to 99999.9999
- ▶ **Q5 Workpiece surface coordinate?** (absolute): Absolute coordinate of the top surface of the workpiece Input range -99999.9999 to 99999.9999
- ▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q7 Clearance height?** (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
- ▶ **Q8 Inside corner radius?**: 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
- ▶ **Q9 Direction of rotation?** $cw = -1$: Machining direction for pockets
 - $Q9 = -1$ up-cut milling for pocket and island
 - $Q9 = +1$ climb milling for pocket and island



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	;ROTATIONAL DIRECTION

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

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

Cycle run

You use Cycle 21 PILOT DRILLING if you subsequently do not use a center-cut end mill (ISO 1641) for clearing out your contour. 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: $t = 0.6 \text{ mm}$
 - At a total hole depth exceeding 30 mm: $t = \text{hole depth} / 50$
 - 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.

Fixed Cycles: Contour Pocket

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

Please note while programming:



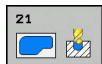
When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **TOOL CALL** block.

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.

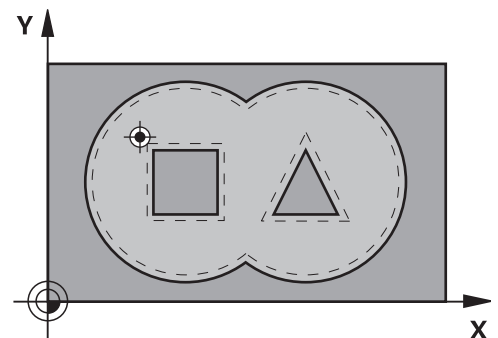
If Q13=0, the TNC uses the data of the tool that is currently in the spindle.

After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the ConfigDatum > CfgGeoCycle > posAfterContPocket parameter to ToolAxClearanceHeight.

Cycle parameters



- ▶ **Q10 Plunging depth?** (incremental): Dimension by which the tool drills in each infeed (minus sign for negative working direction). Input range -99999.9999 to 99999.9999
- ▶ **Q11 Feed rate for plunging?:** Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q13 Rough-out tool number/name?** or QS13: Number or name of rough-out tool. You are able to apply the tool via soft key directly from the tool table.



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

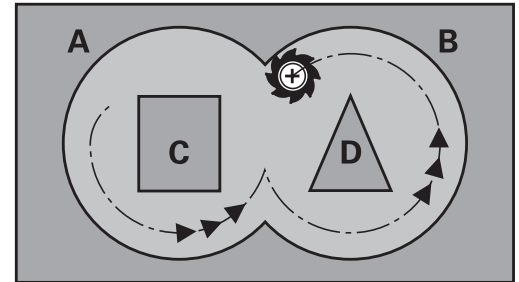
Use Cycle 22 ROUGHING to define the technology data for roughing.

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.



Fixed Cycles: Contour Pocket

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

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.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.



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**. After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the ConfigDatum > CfgGeoCycle > posAfterContPocket parameter to ToolAxClearanceHeight.

Cycle parameters



- ▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- ▶ **Q11 Feed rate for plunging?:** Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q12 Feed rate for roughing?:** Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the TNC has already coarse-roughed the contour. You are able to apply the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the soft key **TOOL NAME**. 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. The TNC will otherwise generate an error message. Input range 0 to 99999 if a number is entered; maximum 16 characters if a name is entered.
- ▶ **Q19 Feed rate for reciprocation?:** Traversing speed of the tool in mm/min during reciprocating plunge-cut. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q208 Feed rate for retraction?:** 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 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 F. ROUGHNG
Q18=1	;COARSE ROUGHING TOOL
Q19=150	;FEED RATE FOR RECIP.
Q208=9999	;RETRACTION FEED RATE
Q401=80	;FEED RATE FACTOR
Q404=0	;FINE ROUGH STRATEGY

- ▶ **Q401 Feed rate factor in %?**: 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
- ▶ **Q404 Fine roughing strategy (0/1)?**: 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.

Fixed Cycles: Contour Pocket

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

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.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

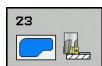


Danger of collision!

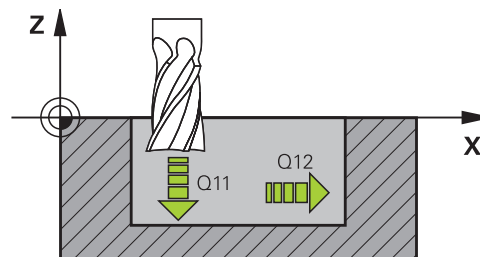
After executing an SL cycle you must program the first traverse motion in the working plane with both coordinate data, e.g. **L X+80 Y+0 R0 FMAX**.

After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the ConfigDatum > CfgGeoCycle > posAfterContPocket parameter to ToolAxClearanceHeight.

Cycle parameters



- ▶ **Q11 Feed rate for plunging?:** Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q12 Feed rate for roughing?:** Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q208 Feed rate for retraction?:** 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 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 F.
ROUGHNG**

**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.

Fixed Cycles: Contour Pocket

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

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.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.



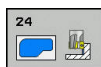
Danger of collision!

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**.

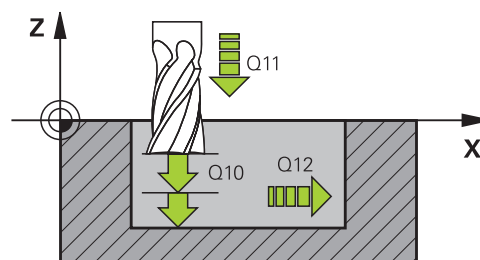
After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the ConfigDatum > CfgGeoCycle > posAfterContPocket parameter to ToolAxClearanceHeight.

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

Cycle parameters



- ▶ **Q9 Direction of rotation?** $cw = -1$: Machining direction:
+1: Rotation counterclockwise
-1: Rotation clockwise
- ▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- ▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q14 Finishing allowance for side?** (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	;ROTATIONAL DIRECTION
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q14=+0	;ALLOWANCE FOR SIDE

Fixed Cycles: Contour Pocket

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

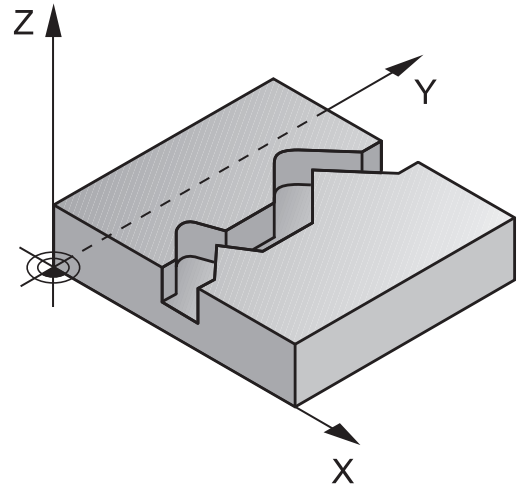
7.9 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125)

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 sub program allows no **APPR-** or **DEP** motions.

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

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

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

**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

- ▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range -99999.9999 to 99999.9999
- ▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range -99999.9999 to 99999.9999
- ▶ **Q5 Workpiece surface coordinate?** (absolute): Absolute coordinate of the top surface of the workpiece Input range -99999.9999 to 99999.9999
- ▶ **Q7 Clearance height?** (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
- ▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- ▶ **Q11 Feed rate for plunging?:** Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q12 Feed rate for roughing?:** Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q15 Climb or up-cut? up-cut = -1:**
Climb milling: Input value = +1
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 F. ROUGHNG
Q15=-1	;CLIMB OR UP-CUT

Fixed Cycles: Contour Pocket

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

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



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

Fixed Cycles: Contour Pocket

7.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)

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,

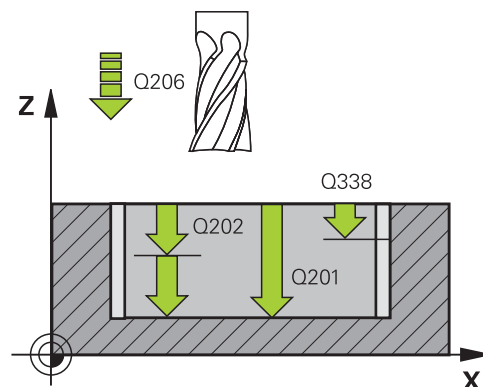
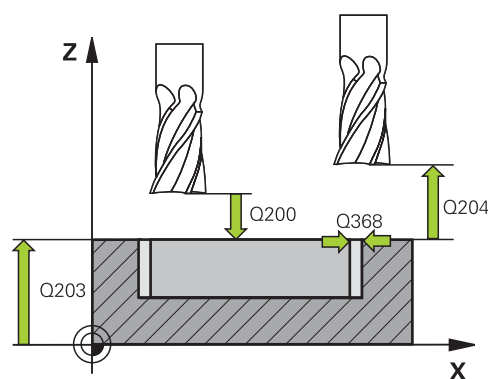
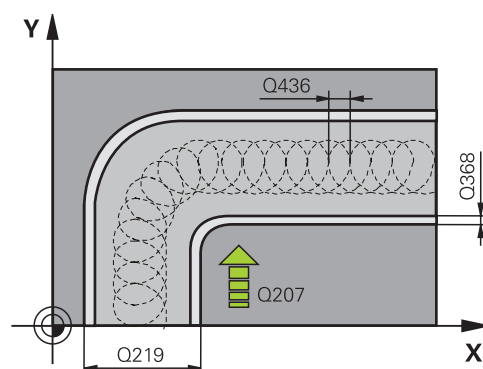
- Do not program positions in incremental dimensions immediately after Cycle 275 since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all principal axes, since the position of the tool at the end of the cycle is not identical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

TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275) 7.11

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2)?**: 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
- ▶ **Q219 Width of slot?** (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
- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane Input range 0 to 99999.9999
- ▶ **Q436 Feed per revolution?** (absolute): Value by which the TNC moves the tool in the machining direction per revolution. Input range 0 to 99999.9999
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: 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 performed)
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot Input range -99999.9999 to 99999.9999
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999



Fixed Cycles: Contour Pocket

7.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)

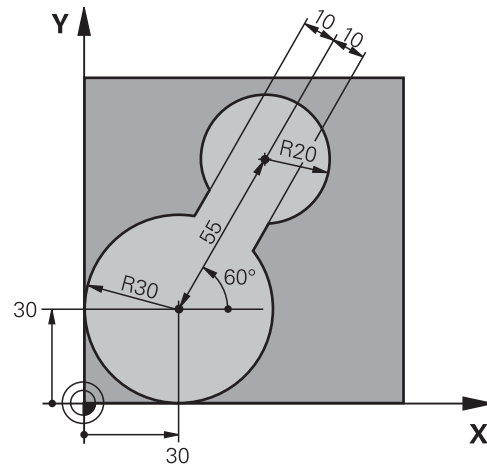
- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Q385 Finishing feed rate?:** Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO, FU, FZ**
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q366 Plunging strategy (0/1/2)?:** 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**
- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor Input range 0 to 99999.9999
- ▶ **Q439 Feed rate reference (0-3)?:** Specify what the programmed feed rate refers to:
0: Feed rate with respect to the tool center point path
1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center point path
2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise it refers to the tool path center
3: Feed rate always refers to the cutting edge

NC blocks

8 CYCL DEF 275 TROCHOIDAL SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q436=2	;INFEE PER REV.
Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEE FOR FINISHING
Q385=500	;FINISHING FEED RATE
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=2	;PLUNGE
Q369=0	;ALLOWANCE FOR FLOOR
Q439=0	;FEED RATE REFERENCE
9 CYCL CALL FMAX M3	

7.12 Programming Examples

Example: Roughing-out and fine-roughing a pocket



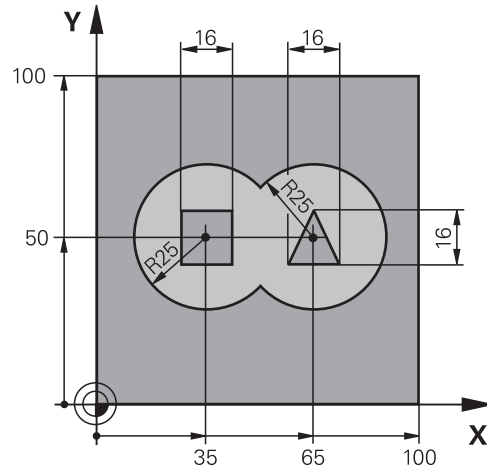
0 BEGIN PGM C20 MM	
1 BLK FORM 0.1 Z X-10 Y-10 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	Definition of workpiece blank
3 TOOL CALL 1 Z S2500	Tool call: coarse roughing tool, diameter 30
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 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 ;ROTATIONAL 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 F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=30000 ;RETRACTION FEED RATE	
9 CYCL CALL M3	Cycle call: Coarse roughing
10 L Z+250 R0 FMAX M6	Tool change

Fixed Cycles: Contour Pocket

7.12 Programming Examples

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 F. ROUGHNG	
Q18=1 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
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



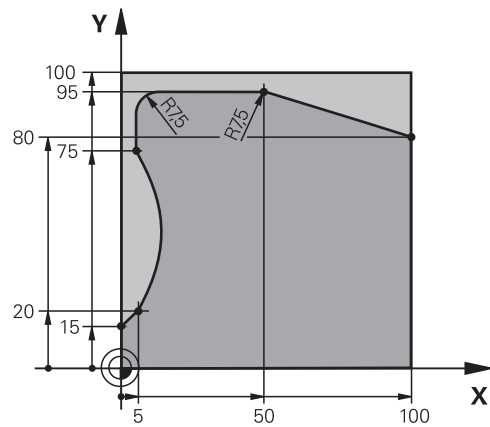
0 BEGIN PGM C21 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: Drill, diameter 12
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4	
7 CYCL DEF 20 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 ;ROTATIONAL 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 F. ROUGHNG	

Fixed Cycles: Contour Pocket

7.12 Programming Examples

Q18=0	;COARSE ROUGHING TOOL	
Q19=150	;FEED RATE FOR RECIP.	
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 F. ROUGHNG	
Q208=30000	;RETRACTION FEED RATE	
15 CYCL CALL		Cycle call: Floor finishing
16 CYCL DEF 24 SIDE FINISHING		Cycle definition: Side finishing
Q9=+1	;ROTATIONAL DIRECTION	
Q10=5	;PLUNGING DEPTH	
Q11=100	;FEED RATE FOR PLNGNG	
Q12=400	;FEED RATE F. ROUGHNG	
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	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 MILLNG	
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 Fixed Cycles: Cylindrical Surface

8.1 Fundamentals

8.1 Fundamentals

Overview of cylindrical surface cycles

Soft key	Cycle	Page
	27 CYLINDER SURFACE	237
	28 CYLINDER SURFACE slot milling	240
	29 CYLINDER SURFACE ridge milling	243
	39 CYLINDER SURFACE Contour	246

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

Cycle run

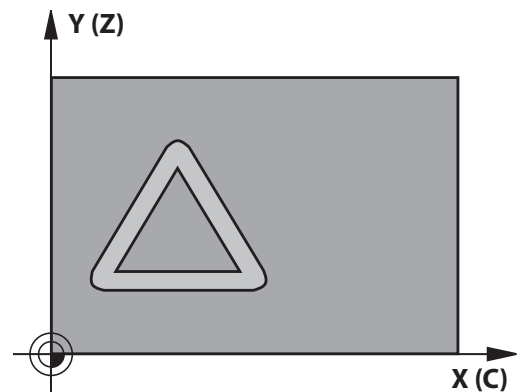
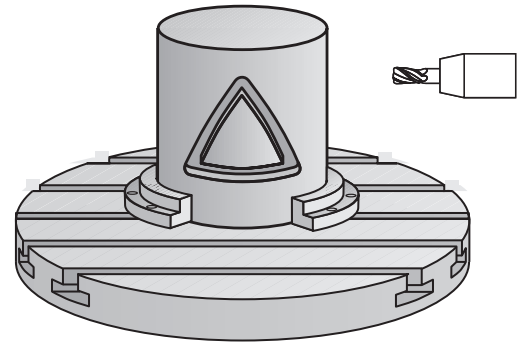
This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle 28 if you want to mill guideways on the cylinder.

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 Subsequently, the tool retracts in the tool axis to the clearance height.



8 Fixed Cycles: Cylindrical Surface

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

Please note while programming:



Refer to your machine manual.

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



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.

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

Cycle parameters



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

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 F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION

8 Fixed Cycles: Cylindrical Surface

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

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

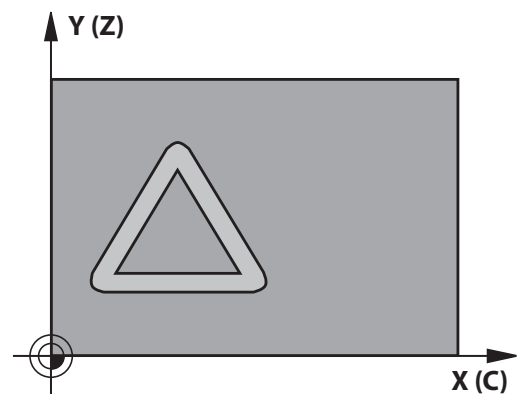
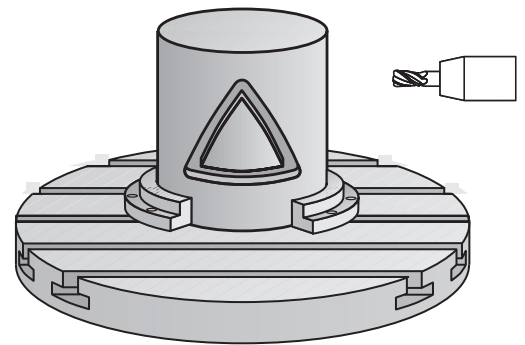
Cycle run

With this cycle you can program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle 27, with this cycle the TNC adjusts 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.



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

8.3

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.



After the end of the cycle, do not use incremental positioning for the tool. Rather, program an absolute position.

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.

8 Fixed Cycles: Cylindrical Surface

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

Cycle parameters



- ▶ **Q1 Milling depth?** (incremental): Distance between cylinder surface and contour bottom. Input range -99999.9999 to 99999.9999
- ▶ **Q3 Finishing allowance for side?** (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
- ▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylinder surface. Input range 0 to 99999.9999
- ▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- ▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**
- ▶ **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- ▶ **Q17 Dimension type? deg=0 MM/INCH=1**: The dimensions for the rotary axis of the subprogram are given either in degrees or in mm/inches
- ▶ **Q20 Slot width?**: Width of the slot to be machined. Input range -99999.9999 to 99999.9999
- ▶ **Q21 Tolerance?**: 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 F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION
Q20=12	;SLOT WIDTH
Q21=0	;TOLERANCE

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

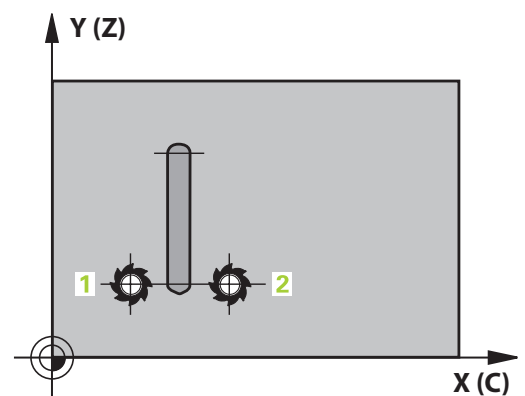
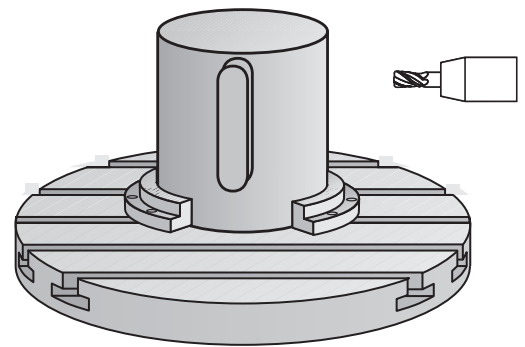
8.4 CYLINDER SURFACE Ridge milling (Cycle 29, DIN/ISO: G129, software option 1)

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.



8 Fixed Cycles: Cylindrical Surface

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

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.

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

Cycle parameters



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

NC blocks

63 CYCL DEF 29 CYL 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 F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION
Q20=12	;RIDGE WIDTH

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

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

Cycle run

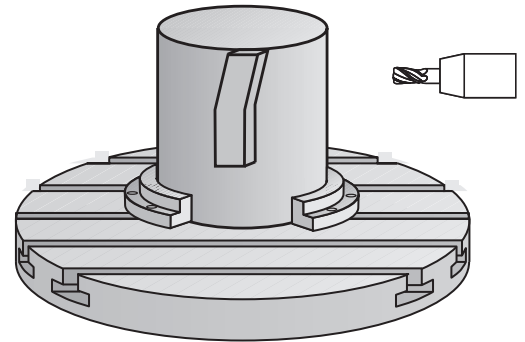
This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle the TNC adjusts the tool so that, with radius compensation active, the wall of the open contour is always parallel to the cylinder axis.

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.



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

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.

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

Cycle parameters



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

NC blocks

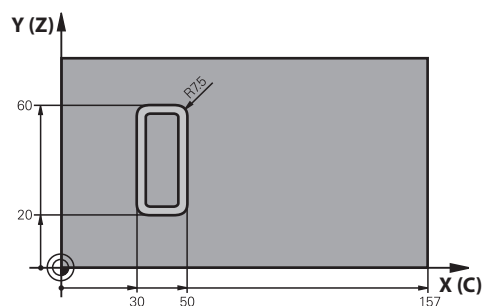
63 CYCL DEF 39 CYL. SURFACE CONTOUR	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION

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	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 F. ROUGHNG	
Q16=25 ;RADIUS	
Q17=1 ;TYPE OF DIMENSION	
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 RN R7.5	
18 L IX-20	
19 RND R7.5	
20 L Y+20	

8

Fixed Cycles: Cylindrical Surface

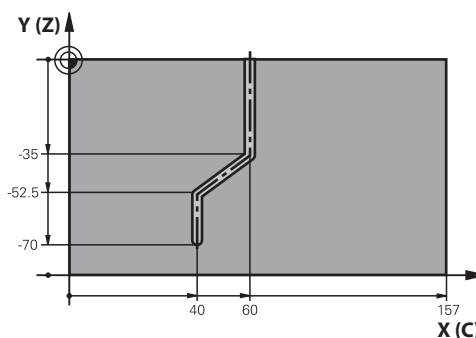
8.6 Programming Examples

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	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 F. ROUGHNG	
Q16=25 ;RADIUS	
Q17=1 ;TYPE OF DIMENSION	
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**

Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

9.1 SL cycles with complex contour formula

Fundamentals

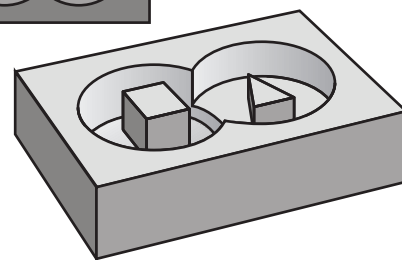
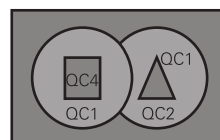
SL cycles and the complex contour formula enable you to form complex contours by combining subcontours (pockets or islands). You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the complete contour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

SL cycles with complex contour formula 9.1

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 CIRCLE1 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM CIRCLE1 MM
```




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

Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

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:




-  ▶ Show the soft-key row with special functions
-  ▶ Menu for functions: Press the soft key for contour and point machining
-  ▶ Press the **SEL CONTOUR** soft key.
- ▶ Enter the full name of the program with the contour definitions 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):




-  ▶ Show the soft-key row with special functions
-  ▶ Menu for functions: Press the soft key for contour and point machining
-  ▶ 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 descriptions 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.

- 
 - Show the soft-key row with special functions
- 
 - Menu for functions: Press the soft key for contour and point machining
- 
 - Press the **CONTOUR FORMULA** soft key. The TNC then displays the following soft keys:

Soft key	Mathematical function
	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. $QC25 = QC1 \ \backslash \ QC2$
	Parenthesis open e.g. $QC12 = QC1 * (QC2 + QC3)$
	Parenthesis closed e.g. $QC12 = QC1 * (QC2 + QC3)$
	Define single contour e.g. $QC12 = QC1$

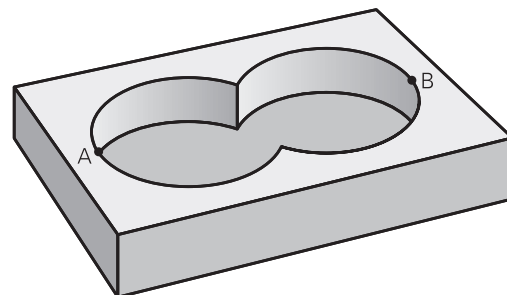
Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

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

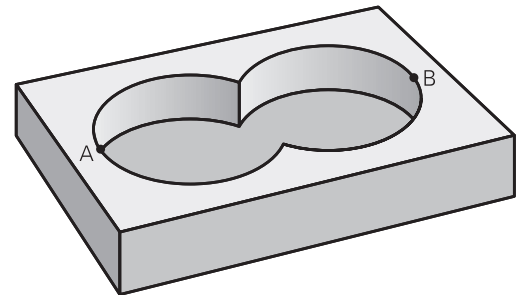
```
0 BEGIN PGM POCKET_B MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET_A MM
```

SL cycles with complex contour formula 9.1

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.



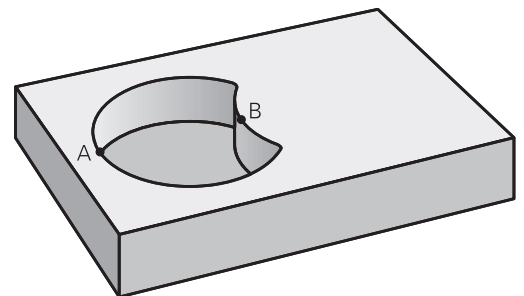
Contour definition program::

```
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 ...
```

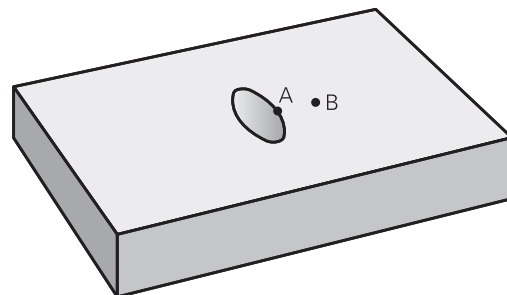
Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

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 ...
```

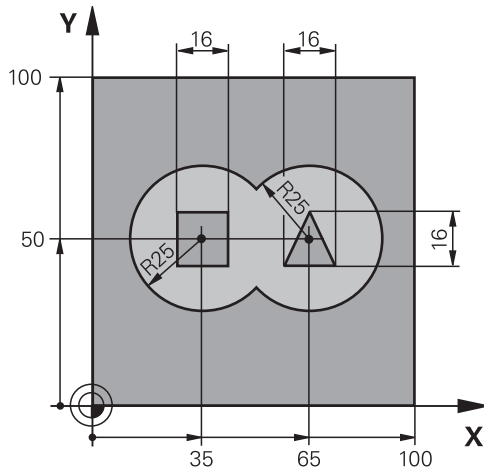
```
56 ...
```

Contour machining with SL Cycles



The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", page 203).

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 ;ROTATIONAL DIRECTION	

Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

9 CYCL DEF 22 ROUGH-OUT	Cycle definition: Rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
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 F. ROUGHNG	
13 CYCL CALL M3	Cycle call: Floor finishing
14 CYCL DEF 24 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;ROTATIONAL DIRECTION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE F. ROUGHNG	
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	

SL cycles with complex contour formula 9.1

Contour description programs:

0 BEGIN PGM CIRCLE1 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 CIRCLE1 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	

Fixed Cycles: Contour Pocket with Contour Formula

9.2 SL cycles with simple contour formula

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.

Fixed Cycles: Contour Pocket with Contour Formula

9.2 SL cycles with simple contour formula

Entering a simple 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

- Menu for functions: Press the soft key for contour and point machining

CONTOUR
DEF

- 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.

ISLAND


- 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 203).

10

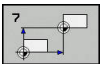

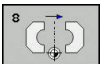
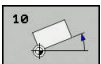
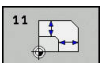
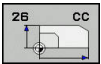
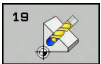
**Cycles: Coordinate
Transformations**

10.1 Fundamentals

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:

Soft key	Cycle	Page
	7 DATUM For shifting contours directly within the program or from datum tables	269
	247 DATUM SETTING Datum setting during program run	275
	8 MIRRORING Mirroring contours	276
	10 ROTATION Rotating contours in the working plane	278
	11 SCALING FACTOR Increasing or reducing the size of contours	280
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	281
	19 WORKING PLANE Machining in tilted coordinate system on machines with swivel heads and/or rotary tables	283

Effectiveness 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.

Reset coordinate transformation:

- 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)

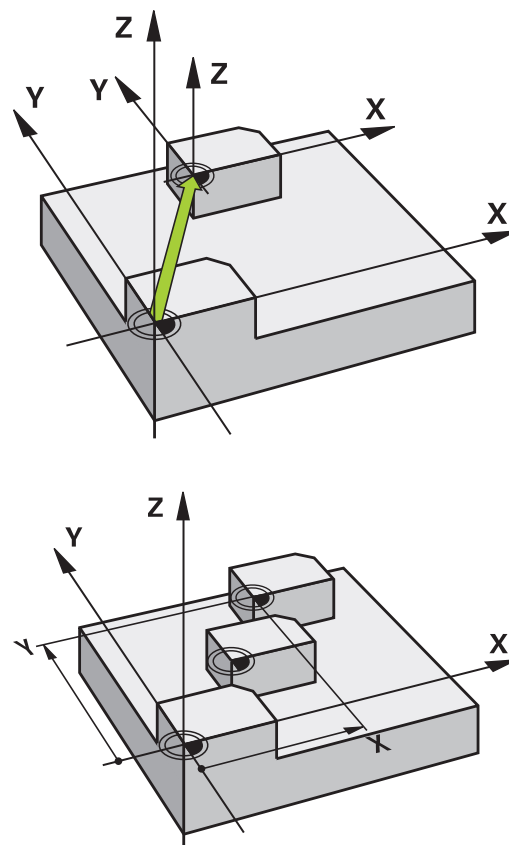
Effect

A datum shift allows machining operations to be repeated at various locations on the workpiece.

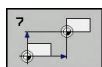
When the datum shift cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

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



- **Displacement:** 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 SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 7.3 Z-5

Cycles: Coordinate Transformations

10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

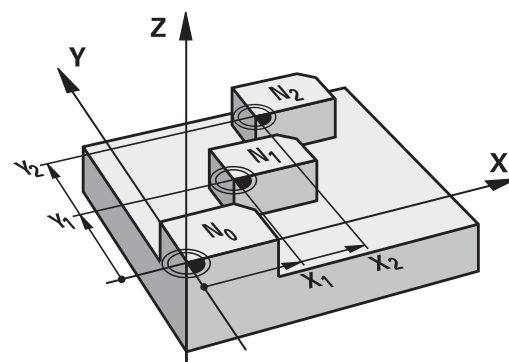
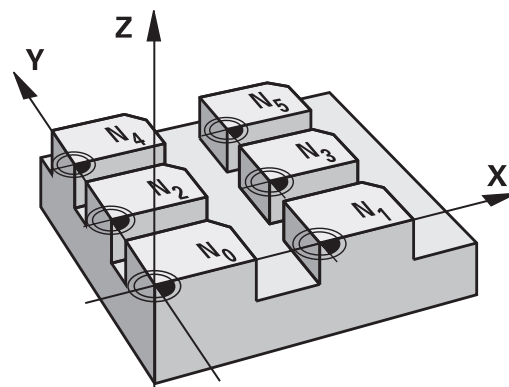
10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

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

DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53) 10.3

Please note while programming:



Danger of collision!

Datums from a datum table are **always and exclusively** referenced to the current reference point (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).

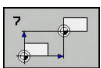
- Use the file management to select the desired table for a test run in the **Test run** operating mode: The table receives the status S
- Use the file management in the **Program run, single block** and **Program run, full sequence** operating modes to select the desired table for program run: The table 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



- **Displacement:** 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

Cycles: Coordinate Transformations

10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

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

- ▶ To 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**.

Editing the datum table in the Programming mode of operation




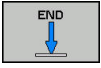


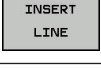
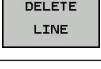
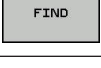
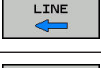
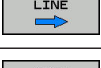
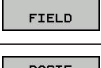
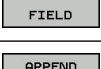
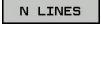
After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise the change might not be included during program run.

Select the datum table in the **Programming** mode of operation **Programming**

PGM
MGT

- ▶ To 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:

DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53) 10.3

Soft key	Function
	Select beginning of table
	Select the table end
	Go to previous page
	Go to next page
	Insert line (only possible at the end of table)
	Delete line
	Find
	Go to beginning of line
	Go to end of line
	Copy the current value
	Insert the copied value
	Add the entered number of lines (datums) to the end of the table

Cycles: Coordinate Transformations

10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

Configuring a 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.

D	X	Y	Z	A	B	C	U
0	141.374	50.002	0	0.0	0.0	0.0	0
1	266.524	50.002	0	0.0	0.0	0.0	0
2	389.881	49.998	0	0.0	0.0	0.0	0
3	469.994	50.001	0	0.0	0.0	0.0	0
4	0.0	0.0	0.0	0.0	0.0	0.0	0
5	0.0	0.0	0.0	0.0	0.0	0.0	0
6	0.0	0.0	0.0	0.0	0.0	0.0	0
7	0.0	0.0	0.0	0.0	0.0	0.0	0
8	0.0	0.0	0.0	0.0	0.0	0.0	0
9	0.0	0.0	0.0	0.0	0.0	0.0	0
10	0.0	0.0	0.0	0.0	0.0	0.0	0
11	0.0	0.0	0.0	0.0	0.0	0.0	0
12	0.0	0.0	0.0	0.0	0.0	0.0	0
13	0.0	0.0	0.0	0.0	0.0	0.0	0
14	0.0	0.0	0.0	0.0	0.0	0.0	0
15	0.0	0.0	0.0	0.0	0.0	0.0	0
16	0.0	0.0	0.0	0.0	0.0	0.0	0
17	0.0	0.0	0.0	0.0	0.0	0.0	0
18	0.0	0.0	0.0	0.0	0.0	0.0	0
19	0.0	0.0	0.0	0.0	0.0	0.0	0
20	0.0	0.0	0.0	0.0	0.0	0.0	0
21	0.0	0.0	0.0	0.0	0.0	0.0	0
22	0.0	0.0	0.0	0.0	0.0	0.0	0
23	0.0	0.0	0.0	0.0	0.0	0.0	0
24	0.0	0.0	0.0	0.0	0.0	0.0	0
25	0.0	0.0	0.0	0.0	0.0	0.0	0
26	0.0	0.0	0.0	0.0	0.0	0.0	0

Leaving 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)

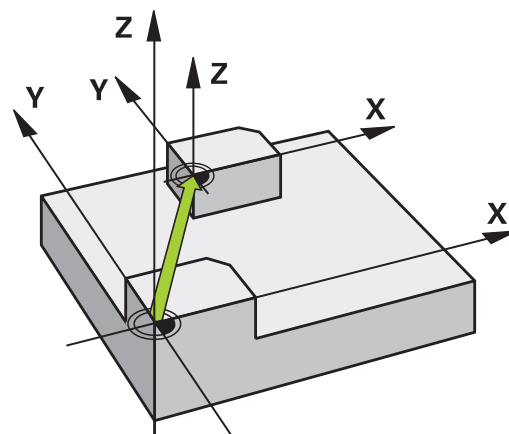
Effect

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 **Electronic handwheel** operating mode. Cycle 247 is not functional in **Test run** mode.

Cycle parameters



- **Number for datum?:** Enter the number of the desired datum from the preset table. Optionally you can use the soft key **SELECT** and select the desired datum 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 (**STATUS POS.**) the TNC shows the active preset number behind the **Datum** dialog.

Cycles: Coordinate Transformations

10.5 MIRRORING (Cycle 8, DIN/ISO: G28)

10.5 MIRRORING (Cycle 8, DIN/ISO: G28)

Effect

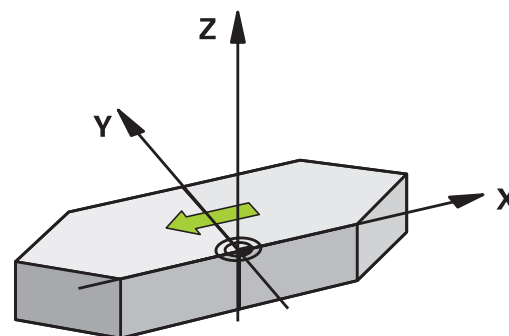
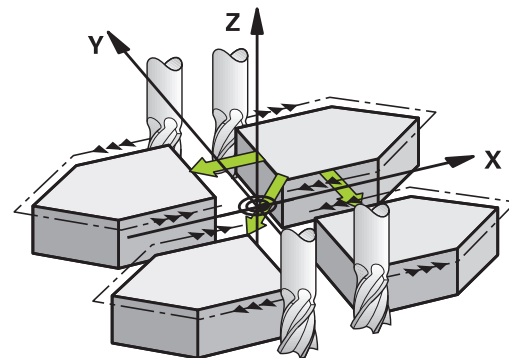
The TNC can machine the mirror image of a contour in the working plane.

The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in the **Positioning with manl.data input** 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.



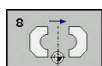
Resetting

Program the MIRROR IMAGE cycle once again with **NO ENT**.

Please note while programming:

If you work in a tilted system with Cycle 8 the following procedure is recommended:

- **First** program the tilting movement and **then** call Cycle 8 MIRRORING!

Cycle parameters

- **Mirror image 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 MIRRORING

80 CYCL DEF 8.1 X Y Z

Cycles: Coordinate Transformations

10.6 ROTATION (Cycle 10, DIN/ISO: G73)

10.6 ROTATION (Cycle 10, DIN/ISO: G73)

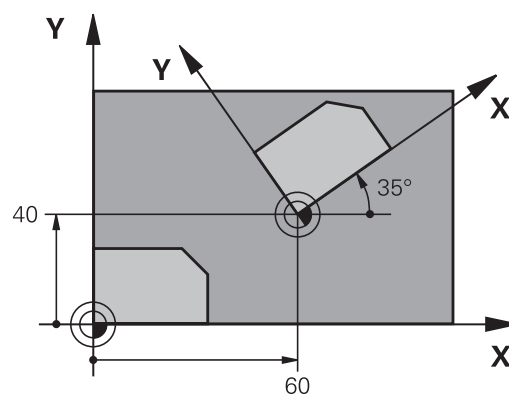
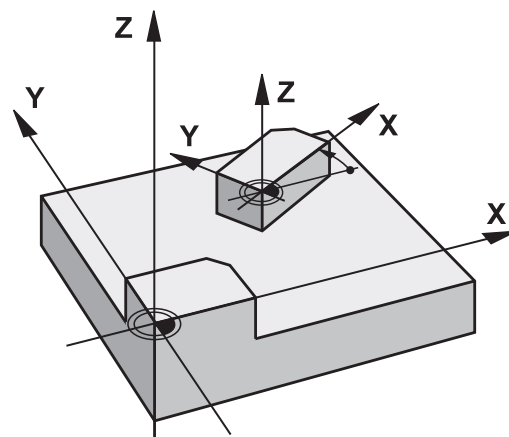
Effect

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



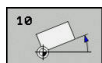
Resetting

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 $+360.000^{\circ}$ (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

Cycles: Coordinate Transformations

10.7 SCALING (Cycle 11, DIN/ISO: G72)

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 manl.data input** 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

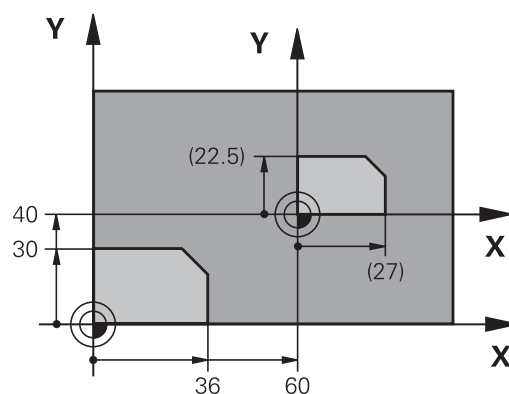
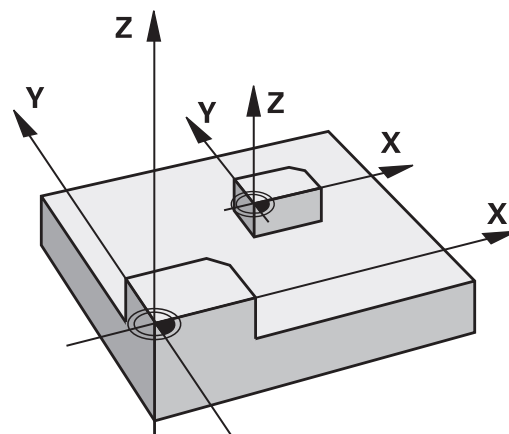
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

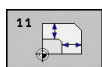
Reduction: SCL less than 1 (down to 0.000 001)

Resetting

Program the SCALING cycle once again with a scaling factor of 1.



Cycle parameters



- **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 SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING FACTOR
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1

10.8 AXIS-SPECIFIC SCALING (Cycle 26)

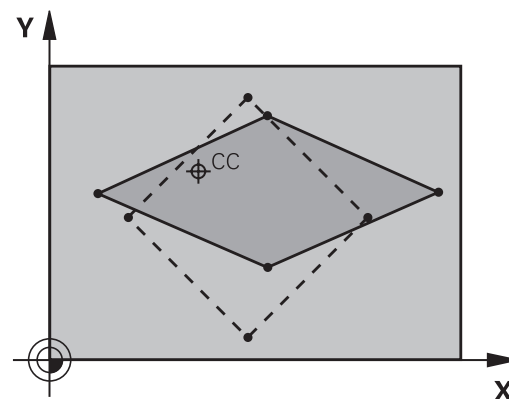
Effect

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 manl.data input** 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.

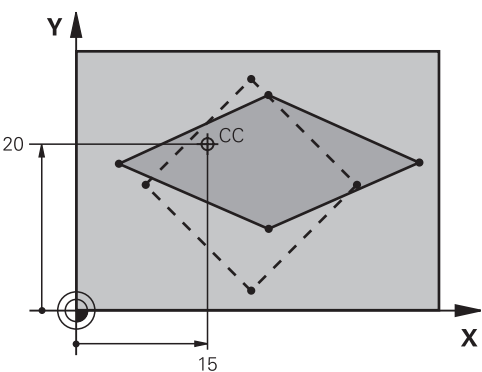
The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING FACTOR) with reference to the active datum.

10.8 AXIS-SPECIFIC SCALING (Cycle 26)

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.



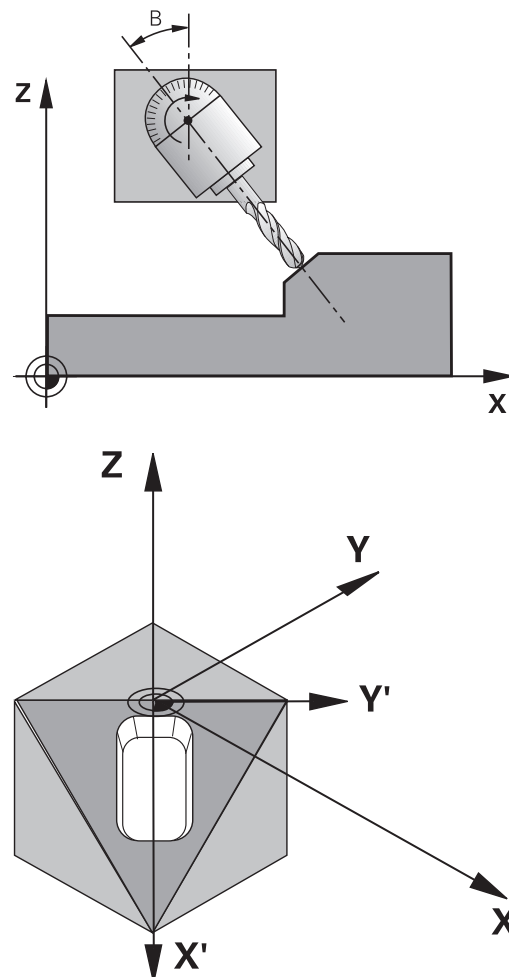
Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

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 current 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.



Cycles: Coordinate Transformations

10.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, software option 1)

Please note while programming:



The functions for tilting the working plane are interfaced to the control 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.

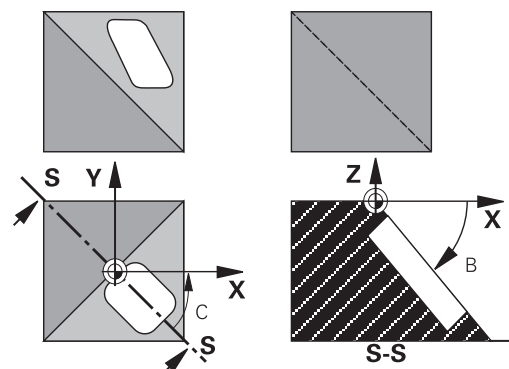
Cycle parameters



- ▶ **Rotary axis and 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=:** Traversing speed of the rotary axis during automatic positioning. Input range 0 to 99999.999
- ▶ **Set-up clearance?** (incremental): 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 reset the tilt angles, 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



Refer to your machine manual.

The machine tool builder determines whether Cycle 19 positions the axes of rotation automatically or whether they must be positioned manually in the program.

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

Cycles: Coordinate Transformations

10.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, software option 1)

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 ABST50	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 a 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.

Monitoring of the working space

The TNC checks 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:

- 1st Activate datum shift
- 2nd Activate tilting function
- 3rd Activate rotation
- ...
- Workpiece machining
- ...
- 1st Reset rotation
- 2nd Reset tilting function
- 3rd Reset datum shift

Cycles: Coordinate Transformations

10.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, software option 1)

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 tilt 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

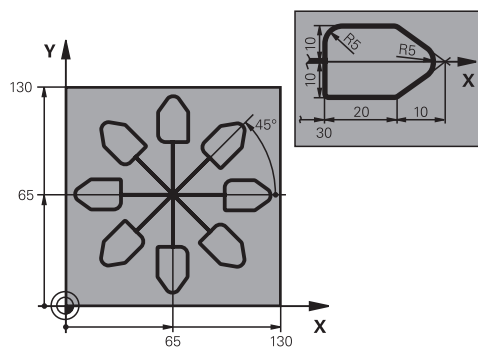
Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each rotary axis in the menu.

10.10 Programming Examples

Example: Coordinate transformation cycles

Program run

- 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 X+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	

10

Cycles: Coordinate Transformations

10.10 Programming Examples

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**










Cycles: Special Functions

11.1 Fundamentals

11.1 Fundamentals

Overview

The TNC provides the following cycles for the following special purposes:

Soft key	Cycle	Page
	9 DWELL TIME	293
	12 Program call	294
	13 Oriented spindle stop	296
	32 TOLERANCE	297
	225 ENGRAVING of texts	318
	291 COUPLING TURNING INTERPOLATION	311
	292 CONTOUR TURNING INTERPOLATION	300
	232 FACE MILLING	323
	239 ASCERTAIN THE LOAD	328

11.2 DWEELL 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 **DWEELL 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 DWEELL TIME

90 CYCL DEF 9.1 DWEELL 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

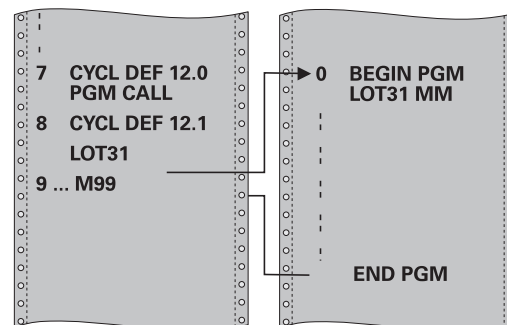
Cycles: Special Functions

11.3 PROGRAM CALL (Cycle 12, DIN/ISO: G39)

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.

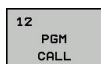
If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

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.

PROGRAM CALL (Cycle 12, DIN/ISO: G39) 11.3

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 DE 12.1 PGM TNC:
 \KLAR35\FK1\50.H

57 L X+20 Y+50 FMAX M99

Cycles: Special Functions

11.4 SPINDLE ORIENTATION (Cycle 13, DIN/ISO: G36)

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 at an angle that has been set by the machine tool builder.

More information: machine tool manual.

Please note while programming:

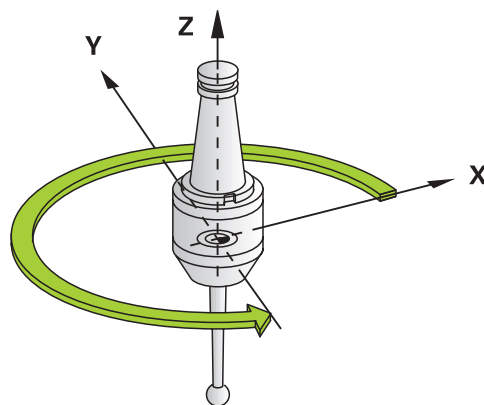


Cycle 13 is used internally for Cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters



- **Angle of orientation:** Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°



NC blocks

93 CYCL DEF 13.0 ORIENTATION

94 CYCL DEF 13.1 ANGLE 180

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.

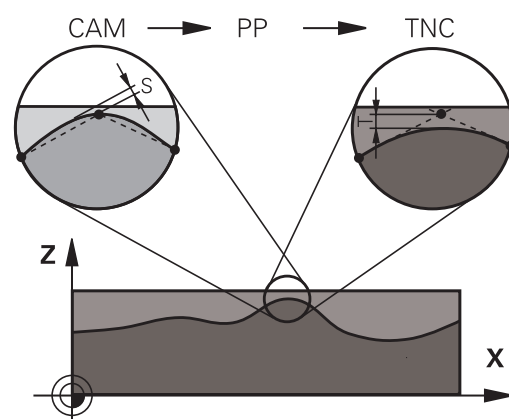
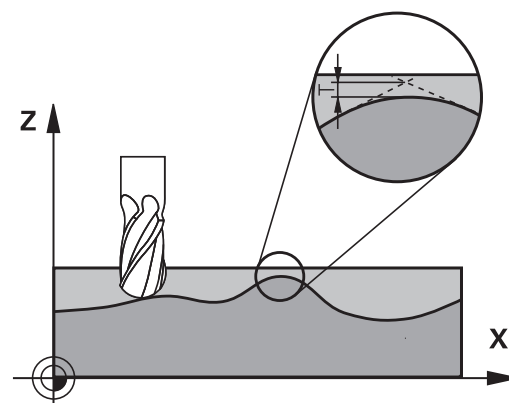
Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**tolerance value**) is set in a machine parameter by the machine manufacturer. With **CYCLE 32** you can change the pre-set tolerance value and select different filter settings, provided 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.



Cycles: Special Functions

11.5 TOLERANCE (Cycle 32, DIN/ISO: G62)

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, unless HSC filters are active on your machine (set 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

Cycles: Special Functions

11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

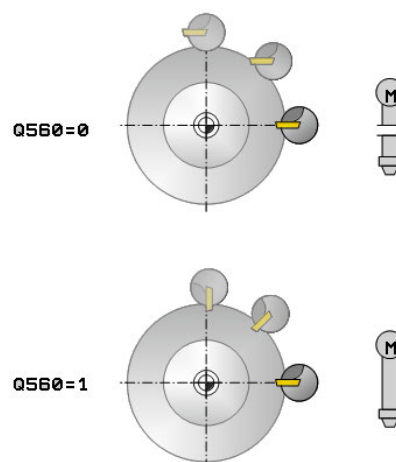
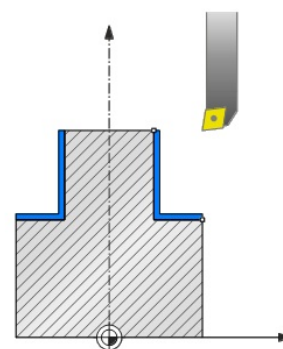
Cycle run

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. If you enter Q560=1, you can turn the contour and the cutting edge is oriented toward the circle center. If you enter Q560=0, you can mill the contour and the spindle is not oriented toward the circle center.

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.



INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, 11.6 DIN/ISO: G292, software option 96)

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.

Cycles: Special Functions

11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

Please note while programming:

A programming example is provided at the end of this chapter, see page 332.



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 contours the TNC checks whether the active tool radius is less than half the diameter at the start of contour Q491 plus the set-up clearance on the side Q357. If this test shows that the tool is too large, the program is canceled.

If Cycle **8 MIRRORING** is active, the TNC does **not** perform the cycle for interpolation turning.

If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the TNC does **not** perform the cycle for interpolation turning.

INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, 11.6 DIN/ISO: G292, software option 96)



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.

Cycles: Special Functions

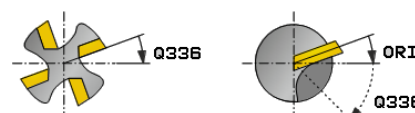
11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

Cycle parameters



- ▶ **Q560 Spindle coupling (0=off, 1=on)?**: Specify whether spindle coupling is executed.
0: Spindle coupling off (mill the contour)
1: Spindle coupling on (turn the contour)
- ▶ **Q336 Angle for spindle orientation?**: The TNC orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input range 0.000 to 360.000
- ▶ **Q546 Reverse tool rotation direction?**: Direction of spindle rotation of the active tool:
3: Tool turns to the right (M3)
4: Tool turns to the left (M4)
- ▶ **Q529 Machining operation (0/1)?**: Specify whether an inside or outside contour is machined:
+1: Inside machining
0: Outside machining
- ▶ **Q221 Oversize for surface?**: Allowance in the working plane. Input range 0 to 99.9999
- ▶ **Q441 Infeed per revolution [mm/rev]?**: Dimension by which the TNC feeds the tool during one revolution. Input range 0.001 to 99.999
- ▶ **Q449 Feed rate / cutting speed? (mm/min)**: Feed rate relative 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 **Q529 MACHINING OPERATION**. From these parameters, the TNC determines the programmed cutting speed at the diameter of the contour starting point.
Q529=1: Feed rate of the tool's center point path is reduced for inside machining
Q529=0: Feed rate of the tool's center point path is increased for outside machining

TO	ORI	P-ANGLE



NC blocks

63 CYCL DEF 292	
CONTOUR.TURNG.INTRP.	
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	;INFED
Q449=2000	;FEED RATE
Q491=0	;CONTOUR START RADIUS
Q357=2	;CLEARANCE TO SIDE
Q445=50	;CLEARANCE HEIGHT

INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, 11.6 DIN/ISO: G292, software option 96)

- ▶ **Q491 Contour starting point (radius)?** (absolute value): Radius of the contour starting point (e.g. X-coordinate, if tool axis is Z). Input range 0.9999 to 99999.9999
- ▶ **Q357 Safety clearance to the side?** (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth Input range 0 to 99999.9
- ▶ **Q445 Clearance height?** (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

Cycles: Special Functions

11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

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.

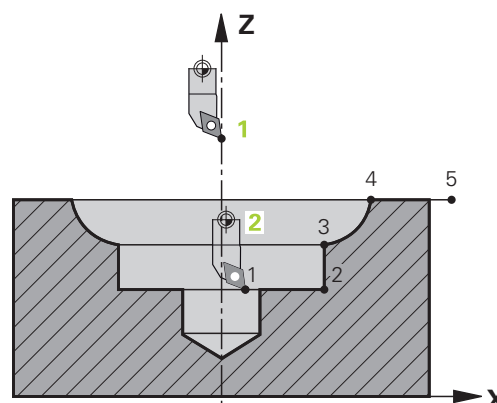
You can use this cycle for inside and outside machining. The following information illustrates some of the notes given in the "Please note while programming" section. You can also find a programming example in "Example: Interpolation Turning Cycle 292", page 332

Inside machining

- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation!** Keep this in mind while 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.



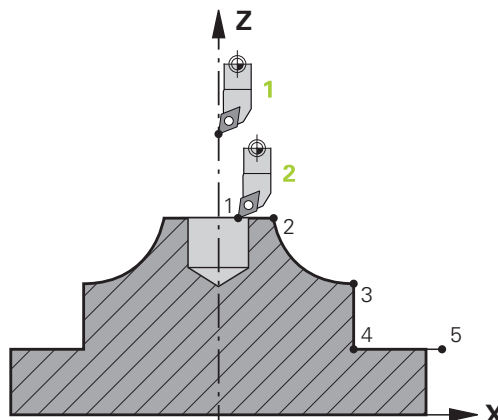
INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, 11.6 DIN/ISO: G292, software option 96)

Outside machining

- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while 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.



Cycles: Special Functions

11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

Defining the tool

Overview

Depending on the entry for parameter Q560 you can either 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 your turning tool as a milling cutter in the tool table (tool.t). In this case the following data from the tool table is taken into account (including delta values): length (L), radius (R), and corner 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 equals Q336, and for inside machining the spindle orientation equals 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.

INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, 11.6 DIN/ISO: G292, software option 96)

■ Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)

You can perform interpolation turning with a milling tool. In this case the following data from the tool table is taken into account (including delta values): length (L), radius (R), and corner 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 equals Q336, and for inside machining the spindle orientation equals 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

Cycles: Special Functions

11.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ISO: G292, software option 96)

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 contours the TNC checks whether the active tool radius is less than half the diameter at the start of contour Q491 plus the set-up clearance on the side Q357. If this test shows that the tool is too large, the program is canceled.



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

COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, 11.7 software option 96)

11.7 COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)

Cycle run

Cycle **291 COUPLG.TURNG.INTERP.** couples the tool spindle to the position of the linear axes, or cancels this spindle coupling. With 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 COUPLG.TURNG.INTERP.** is run in milling mode and is CALL-active.

Cycle run if Q560=1:

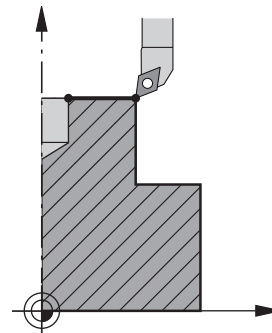
- 1 The TNC first stops the spindle (M5).
- 2 The TNC orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation Q336 is taken into account. If an "ORI" value is given in the tool table, it is also taken into account if so defined.
- 3 The tool spindle is 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. (With Cycle 291 and end of program/internal stop resp.)

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 and **CYCLE CALL** program the operation you wish to perform. In order to describe the circular motions of the linear axes linear/polar coordinates are used, for example. A programming example is provided at the end of this section, see page 330.



Cycles: Special Functions

11.7 COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)



Cycle 291 is CALL-active.

Programming of M3/M4 is not required. In order to describe the circular motions of the linear axes, you can use **CC** and **C** coordinates, for example.

If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter Q561=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With Q561=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program Q561=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

If you programmed parameter Q561=1, you must program the following in order to complete the operation of interpolation turning:

- R0, overrides radius compensation
- Cycle 291 with parameters Q560=0 and Q561=0, overrides spindle coupling
- **CYCLE CALL**, for calling Cycle 291
- **TOOL CALL** overrides the conversion of parameter Q561

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** perform the cycle for interpolation turning.

If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the TNC does **not** perform the cycle for interpolation turning.



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.

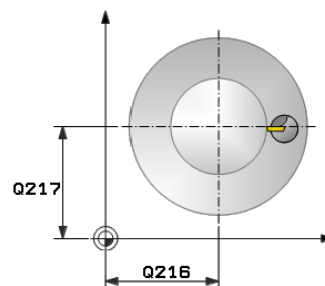
Software option 96 must be enabled.

COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, 11.7 software option 96)

Cycle parameters



- ▶ **Q560 Spindle coupling (0=off, 1=on)?**: Specify whether the tool spindle is coupled to the position of the linear axes. If spindle coupling is active, the tool's cutting edge is oriented to the center of turning.
0: Spindle coupling off
1: Spindle coupling on
- ▶ **Q336 Angle for spindle orientation?**: The TNC orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input range 0.000 to 360.000
- ▶ **Q216 Center in 1st axis?** (absolute): Center of turning in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q217 Center in 2nd axis?** (absolute): Center of turning in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q561 Convert from turning tool (0/1)**: Only relevant if you define the turning tool in the turning tool table (toolturn.trn). With this parameter you decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool.
0: No change; the turning tool is interpreted as it described in the turning tool table (toolturn.trn). In this case you may not use radius compensation **RR** or **RL**. Furthermore, you must describe the movement of the tool center path **TCP** without spindle coupling when programming. This kind of programming is much more difficult.
1: The value XL from the turning tool table (toolturn.trn) will be interpreted as a radius R from a milling tool table. This makes it possible to use radius compensation **RR** and **RL** when programming your contour. This kind of programming is recommended.



NC blocks

64 CYCL DEF 291 COUPLG.TURNG.INTERP.	
Q560=1	;SPINDLE COUPLING
Q336=0	;ANGLE OF SPINDLE
Q216=50	;CENTER IN 1ST AXIS
Q217=50	;CENTER IN 2ND AXIS
Q561=1	;TURNING TOOL CONVERSION

Cycles: Special Functions

11.7 COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)

Defining the tool

Overview

Depending on the entry for parameter Q560 you can either activate (Q560=1) or deactivate (Q560=0) coupling for the interpolation turning 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)

COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, 11.7 software option 96)

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 your turning tool as a milling cutter in the tool table (tool.t). In this case the following data from the tool table is taken into account (including delta values): length (L), radius (R), and corner 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 equals Q336, and for inside machining the spindle orientation equals $Q336+180$.



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 perform interpolation turning with a milling tool. In this case the following data from the tool table is taken into account (including delta values): length (L), radius (R), and corner 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 equals Q336, and for inside machining the spindle orientation equals $Q336+180$.

■ Define a turning tool in the turning tool table (toolturn.trn)

If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case the orientation of the spindle to the center of turning takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table), parameter Q336, and parameter Q561.

Cycles: Special Functions

11.7 COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)



If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter Q561=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With Q561=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program Q561=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

If you programmed parameter Q561=1, you must program the following in order to complete the operation of interpolation turning:

- R0, overrides radius compensation
- Cycle 291 with parameters Q560=0 and Q561=0, overrides spindle coupling
- **CYCLE CALL**, for calling Cycle 291
- **TOOL CALL** overrides the conversion of parameter Q561

If you programmed parameter Q561=1, you may only use the following types of tools:

- TYPE: ROUGH, FINISH, BUTTON with the machining directions TO: 1 or 8, XL>=0
- TYPE: ROUGH, FINISH, BUTTON with the machining directions TO: 7, XL<=0

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	ORI + Q336
Interpolation turning, inside	8	ORI + Q336

COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, 11.7 software option 96)

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1, 7, 8
- TYPE: FINISH, with the machining directions TO: 1, 7, 8
- TYPE: BUTTON, with the machining directions TO: 1, 7, 8



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

Cycles: Special Functions

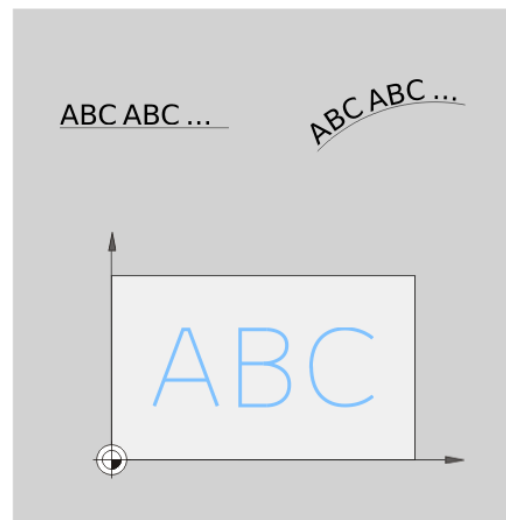
11.8 ENGRAVING (Cycle 225, DIN/ISO: G225)

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.

The text to be engraved can also be transferred with a string variable (**QS**).

Parameter Q347 influences the rotational position of the letters.

If Q374=0° to 180°, the characters are engraved from left to right.

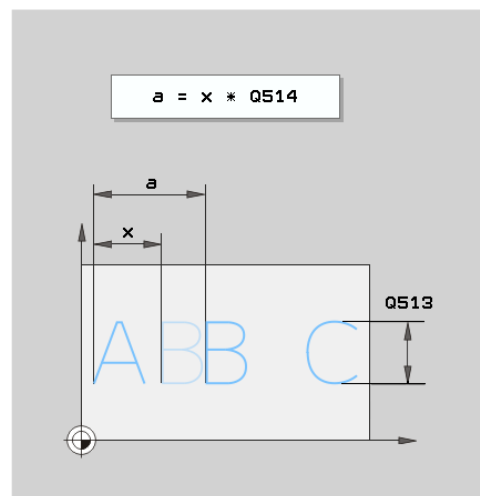
If Q374 is greater than 180°, the direction of engraving is reversed.

When engraving on a circular arc, the starting point is at bottom left, above the first character to be engraved. (With older software versions there was sometimes a pre-positioning to the center of the circle.)

Cycle parameters



- ▶ **Q500 Engraving text?:** 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 ASCII keyboard represents normal text input. Allowed entry characters: see "Engraving system variables", page 322
- ▶ **Q513 Character height?:** (absolute): Height of the characters to be engraved in mm. Input range 0 to 99999.9999
- ▶ **Q514 Character spacing factor?:** 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
- ▶ **Q515 Font?:** Currently without function
- ▶ **Q516 Text on a line/on an arc(0/1)?:**
Engrave the text in a straight line: Input = 0
Engrave the text on an arc: Input = 1
Engrave the text on an arc, circumferentially (not necessarily legible from below): Input = 2
- ▶ **Q374 Angle of rotation?:** Center angle if the text is to be arranged on an arc. Engraving angle when text is in a straight line. Input range -360.0000 to +360.0000°
- ▶ **Q517 Radius of text on an arc?:** (absolute): Radius of the arc in mm on which the TNC is to arrange the text Input range 0 to 99999.9999
- ▶ **Q207 Feed rate for milling?:** Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and engraving floor
- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999



NC blocks

62 CYCL DEF 225 ENGRAVING	
Q500="A"	;ENGRAVING TEXT
Q513=10	;CHARACTER HEIGHT
Q514=0	;SPACE FACTOR
Q515=0	;FONT
Q516=0	;TEXT ARRANGEMENT
Q374=0	;ANGLE OF ROTATION
Q517=0	;CIRCLE RADIUS
Q207=750	;FEED RATE FOR MILLNG
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
Q367=+0	;TEXT POSITION
Q574=+0	;TEXT LENGTH

11.8 ENGRAVING (Cycle 225, DIN/ISO: G225)

- ▶ **Q204 2nd set-up clearance?** (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**
- ▶ **Q574 Maximum text length?** (mm/inch): Enter the maximum text length here. The TNC also takes into account parameter Q513 Character height. If Q513=0, the TNC engraves the text over exactly the length indicated in parameter Q574, and scales the character height correspondingly. If Q513 is greater than zero, the TNC checks whether the actual text length exceeds the maximum text length entered in Q574. If that is the case, the TNC displays an error message.
- ▶ **Q367 Reference for text position (0-6)?** Enter here the reference for the position of the text. Depending on whether the text is engraved on an arc or a straight line (parameter Q516), the following entries are possible:
If engraved on an arc, the text position refers to the following point:
 - 0 = Center of the circle
 - 1 = Bottom left
 - 2 = Bottom center
 - 3 = Bottom right
 - 4 = Top right
 - 5 = Top center
 - 6 = Top left**If engraved on a straight line, the text position refers to the following point:**
 - 0 = Bottom left
 - 1 = Bottom left
 - 2 = Bottom center
 - 3 = Bottom right
 - 4 = Top right
 - 5 = Top center
 - 6 = Top left

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

Cycles: Special Functions

11.8 ENGRAVING (Cycle 225, DIN/ISO: G225)

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 **SYSSTR ID321** function)



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)

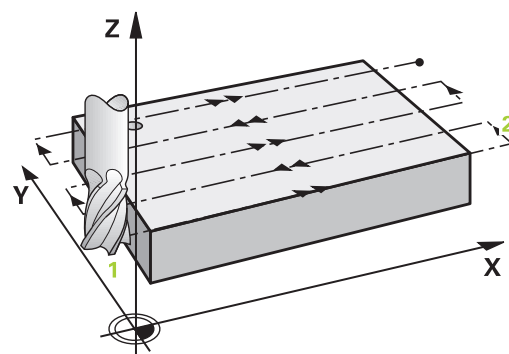
Cycle run

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 end 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 at **FMAX** to the 2nd set-up clearance.



Cycles: Special Functions

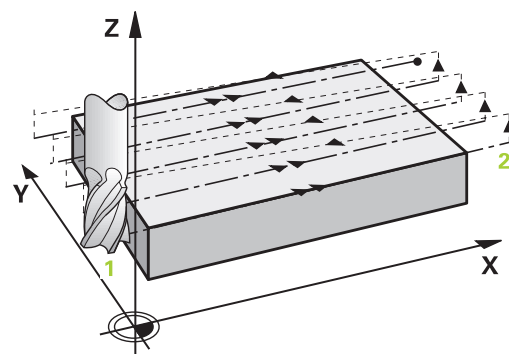
11.9 FACE MILLING (Cycle 232, DIN/ISO: G232)

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 at **FMAX** to the 2nd set-up clearance.

Strategy Q389=2

- 3 The tool subsequently advances to the end point **2** at the programmed feed rate for milling. The end point lies outside the surface. The TNC 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 at **FMAX** to the 2nd set-up clearance.



Please note while programming:

Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

If **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS** are entered as equal values, the TNC does not run the cycle (depth = 0 has been programmed).

Program Q227 greater than Q386. Otherwise, the TNC will display an error message.

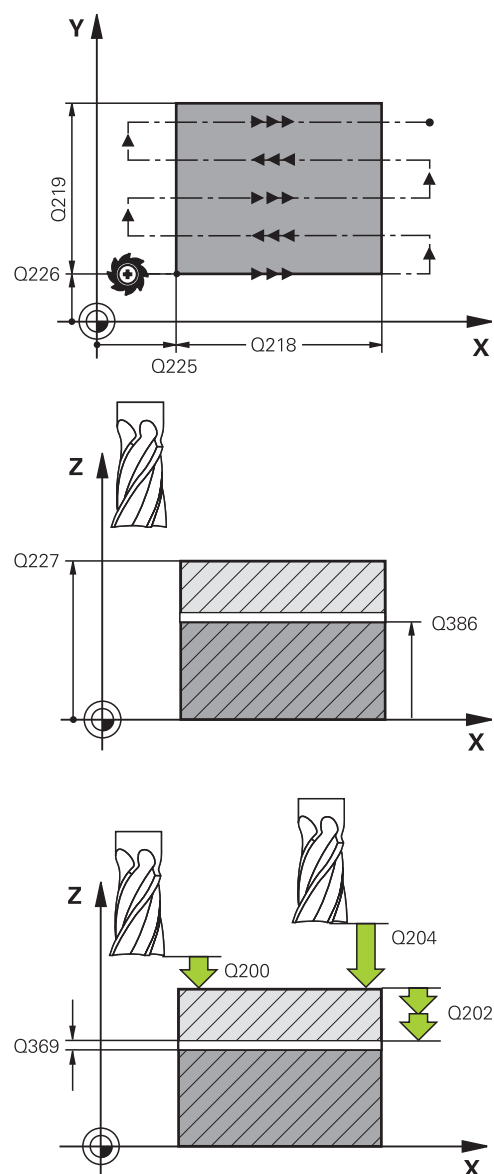
Cycles: Special Functions

11.9 FACE MILLING (Cycle 232, DIN/ISO: G232)

Cycle parameters

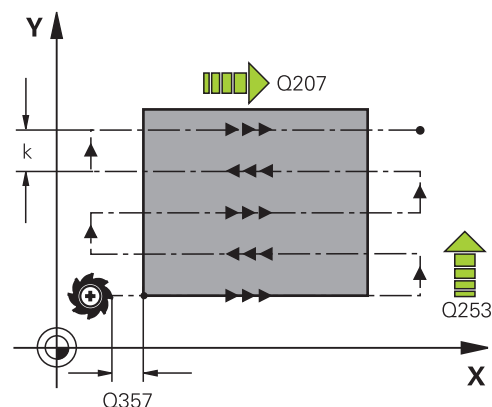


- ▶ **Q389 Machining strategy (0/1/2)?**: Determine how the TNC should machine the surface:
0: Meander machining, stepover at the positioning feed rate outside the surface being machined
1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
2: Line-by-line machining, retraction and stepover at the positioning feed rate
- ▶ **Q225 Starting point in 1st axis?** (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
- ▶ **Q226 Starting point in 2nd axis?** (absolute): Starting point coordinate of the surface to be machined in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999
- ▶ **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis on which the surface is to be face-milled. Input range -99999.9999 to 99999.9999
- ▶ **Q218 First side length?** (incremental): Length of the surface to be machined in the major 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
- ▶ **Q219 Second side length?** (incremental): 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 **STARTNG PNT 2ND AXIS**. Input range -99999.9999 to 99999.9999
- ▶ **Q202 Maximum plunging depth?** (incremental): **Maximum** infeed per cut. 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
- ▶ **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range 0 to 99999.9999



FACE MILLING (Cycle 232, DIN/ISO: G232) 11.9

- ▶ **Q370 Max. path overlap factor?:** 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
- ▶ **Q207 Feed rate for milling?:** Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- ▶ **Q385 Finishing feed rate?:** Traversing speed of the tool in mm/min while milling the last infeed. Input range 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- ▶ **Q253 Feed rate for pre-positioning?:** 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**
- ▶ **Q200 Set-up clearance?** (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
- ▶ **Q357 Safety clearance to the side?** (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
- ▶ **Q204 2nd set-up clearance?** (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. OVERLAP
Q207=500	;FEED RATE FOR MILLNG
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

Cycles: Special Functions

11.10 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143)

11.10 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143)

Cycle run

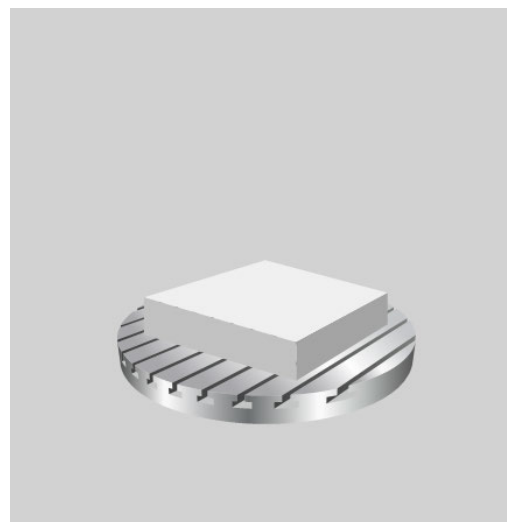
The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of table axes. Option 143 LAC (Load Adaptive Control) and Cycle 239 ASCERTAIN 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.



ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 11.10 143)

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.

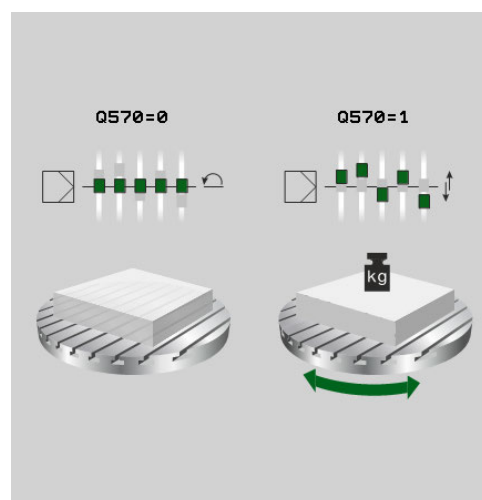
Before cycle start, the TNC might move to a safe position; this position is defined by the machine tool builder!

Before using this cycle, contact your machine tool builder for details on the type and scope of movements performed in Cycle 239!

Cycle parameters



- **Q570 Load (0 = Delete/1 = Ascertain)?**: Specify whether the TNC should perform an LAC (Load adaptive control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters should be reset:
 - 0**: Reset LAC; the values most recently ascertained by the TNC are reset, and the TNC uses load-independent feedforward and controller parameters
 - 1**: Perform a weighing run; the TNC moves the axes and thus ascertains the feedforward and controller parameters depending on the current load. The values ascertained are activated immediately



NC blocks

62 CYCL DEF 239 ASCERTAIN THE LOAD

Q570=+0 ;LOAD ASCERTATION

Cycles: Special Functions

11.11 Programming examples

11.11 Programming examples

Example: Interpolation Turning Cycle 291

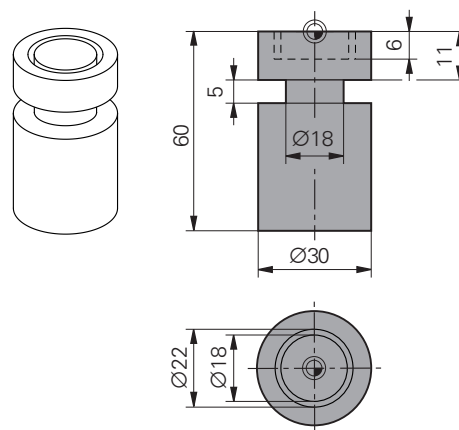
Cycle **291 COUPLG.TURNG.INTERP.** is used in the following program. This programming example illustrates the machining of an axial recess and a radial recess.

Program sequence

- Turning tool as defined in toolturn.trn: tool No 10: TO:1, ORI:0, TYPE:ROUGH, tool for axial recesses
- Turning tool as defined in toolturn.trn: tool No 11: TO:8, ORI:0, TYPE:ROUGH, tool for radial recesses

Program run

- Tool call: tool for axial recess
- Start of interpolation turning: Description and call of Cycle 291; Q560=1
- 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
- End of interpolation turning: Description and call of Cycle 291; Q560=0



By converting parameter Q561, the turning tool is displayed in the simulation graphic as a milling tool.

0 BEGIN PGM 1 MM	
1 BLK FORM CYLINDER Z R15 L60	Definition of workpiece blank: Cylinder
2 TOOL CALL 10	Tool call: 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	
Q561=+1 ;TURNING TOOL CONVERSION	
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

Programming examples 11.11

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	
Q561=+0 ;TURNING TOOL CONVERSION	
18 CYCL CALL	Call the cycle
19 TOOL CALL 11	Tool call: 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	
Q561=+1 ;TURNING TOOL CONVERSION	
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 LP PR+50 FMAX	
40 L 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	

Cycles: Special Functions

11.11 Programming examples

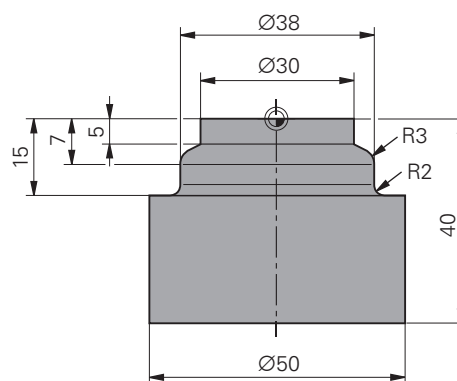
Q561=+0	;TURNING TOOL CONVERSION	
42 CYCL CALL		Call the cycle
43 TOOL CALL 11		Repeated TOOL CALL in order to override the conversion of parameter Q561
44 M30		
45 END PGM 1 MM		

Example: Interpolation Turning Cycle 292

Cycle **292 CONTOUR.TURNG.INTRP.** is used in the following program. This programming example illustrates the machining of an outside contour with the milling spindle rotating.

Program run

- 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 LABEL 1	
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	;INFEEED
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	

Programming examples 11.11

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

Cycles: Turning

12.1 Turning Cycles (software option 50)

12.1 Turning Cycles (software option 50)

Overview

Defining turning cycles:



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

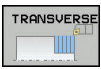



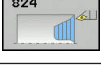

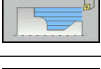

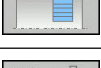







- Menu for cycle group: Press the **TURNING** soft key
- Select cycle group, e.g. cycles for longitudinal turning
- Select cycle, e.g. TURN SHOULDER, LONGITUDINAL

The TNC offers the following cycles for turning operations:

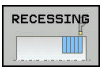
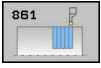




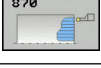




Soft key	Cycle group	Cycle	Page
	Special cycles		
		ADAPT ROTARY COORDINATE SYSTEM(Cycle 800, DIN/ISO: G800)	342
		RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801)	348
		GEAR HOBBING (Cycle 880, DIN/ISO: G880)	449
		CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)	454
	Cycles for longitudinal turning		349
		TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811)	350
		TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812)	353
		TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813)	357
		TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)	360
		TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)	364
		TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)	368

Turning Cycles (software option 50) 12.1

Soft key	Cycle group	Cycle	Page
	Cycles for transverse turning		349
		TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)	372
		TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)	375
		TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)	379
		TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)	382
		TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)	386
		TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)	368
	Cycles for recessing		
		SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)	390
		RADIAL RECESSING EXTENDED (Cycle 842, DIN/ISO: G842)	393
		RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)	398
		SIMPLE AXIAL RECESSING (Cycle 851, DIN/ISO: G851)	402
		AXIAL RECESSING EXTENDED (Cycle 852, DIN/ISO: G852)	405
		AXIAL RECESSING (Cycle 850, DIN/ISO: G850)	410

Cycles: Turning

12.1 Turning Cycles (software option 50)

Soft key	Cycle group	Cycle	Page
	Cycles for recessing		
		RADIAL RECESSING (Cycle 861, DIN/ISO: G861)	414
		RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)	418
		RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)	422
		AXIAL RECESSING (Cycle 871, DIN/ISO: G871)	426
		AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)	429
		AXIAL RECESSING (Cycle 870, DIN/ISO: G870)	434
	Cycles for thread turning		
		THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)	438
		THREAD EXTENDED (Cycle 832, DIN/ISO: G832)	441
		CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ISO: G830)	445

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 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**

12.1 Turning Cycles (software option 50)

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.

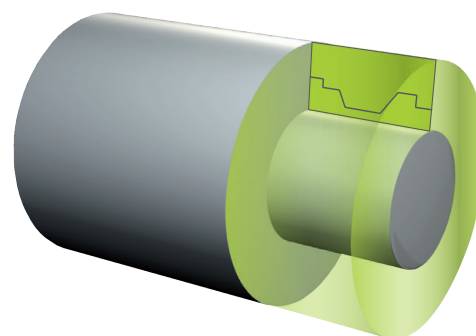
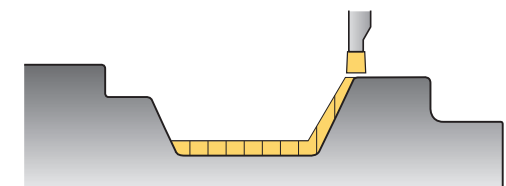
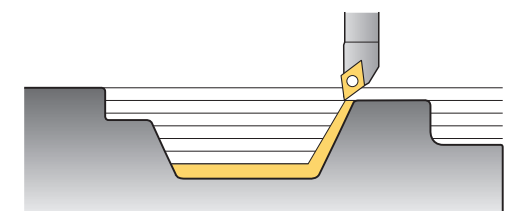
To activate the blank form update, program the **TURNDATA BLANK** function and link to a program or subprogram with a workpiece blank specification. The workpiece blank defined in **TURNDATA BLANK** determines the area to be machined with the blank form update. **TURNDATA BLANK OFF** deactivates blank 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:

Soft key	Workpiece blank definition
	Deactivate blank form update TURNDATA BLANK OFF : No input
	Workpiece blank definition in a program: Enter name of the file
	Workpiece blank definition in a program: Enter the string parameter with the program name
	Workpiece blank definition in a subprogram: Enter the number of the subprogram
	Workpiece blank definition in a subprogram: Enter the name of the subprogram
	Workpiece blank definition in a subprogram: Enter the string parameter with the subprogram name

Activate blank form update and define workpiece blank:

-  ► Show the soft-key row with special functions
-  ► Menu: Press the **TURNING PROGRAM FUNCTIONS** soft key
-  ► Press the **BASIC FUNCTIONS** soft key
-  ► Select the function for blank form update

NC syntax

11 FUNCTION TURNDATABLANK LBL 20

Cycles: Turning

12.2 ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800)

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.

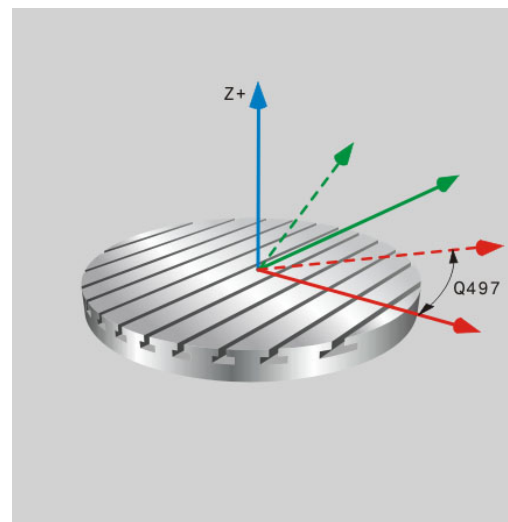
You need to position the tool appropriately with respect to the turning spindle, in order to be able to perform a turning operation. You can use Cycle **800 ADAPT ROTARY COORDINATE SYSTEM** for this.

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 inclined machining, you can use Cycle 800 to orient the coordinate system to the positions 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.



ADAPT ROTARY COORDINATE SYSTEM 12.2 (Cycle 800, DIN/ISO: G800)

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 RO FMAX**.

Cycles: Turning

12.2 ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800)

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.



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

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.



Perform the coupling or decoupling only when the spindle for turning is motionless. 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.

ADAPT ROTARY COORDINATE SYSTEM 12.2 (Cycle 800, DIN/ISO: G800)

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:

- Mirroring of the tool data (Q498 **REVERSE TOOL**) is reset by a **TOOL CALL**.
- The function **ECCENTRIC TURNING** Q535 is reset at the end of the program or if the program is canceled (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 mirror the tool data (**Q498 REVERSE TOOL**) only if 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 parameter **Q530 INCLINED MACHINING**, the TNC activates the function **M144** (also see the "Inclined Turning" User's Manual).

Cycles: Turning

12.2 ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800)

Cycle parameters



- ▶ **Q497 Precession angle?:** Angle to which the TNC aligns the tool. Input range 0 to 359.9999
- ▶ **Q498 Reverse tool (0=no/1=yes)?:** Mirror tool for inside/outside machining. Input range 0 and 1.
- ▶ **Q530 Inclined machining?:** Position the tilting axes for inclined machining:
 - 0:** Maintain position of tilted axes (axis must already have been positioned)
 - 1:** Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between the workpiece and tool 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 later in a separate positioning block (STAY). The TNC stores the position values in the parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis).
- ▶ **Q531 Angle of incidence?:** Angle of incidence for aligning the tool. Input range: -180° to +180°
- ▶ **Q532 Feed rate for positioning?:** Traverse speed of the tilting axis during automatic positioning. Input range 0.001 to 99999.999
- ▶ **Q533 Preferred dir. of incid. angle?:** 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. Using parameter Q533, configure which of the solution options the TNC should apply:
 - 0:** Choose option with shortest way
 - 1:** Choose option with negative direction
 - +1:** Choose option with positive direction
 - 2:** Choose option with negative direction at a range of between -90° and -180°
 - +2:** Choose option with positive direction at a range of between +90° and +180°

ADAPT ROTARY COORDINATE SYSTEM 12.2 (Cycle 800, DIN/ISO: G800)

- ▶ **Q535 Eccentric turning?:** 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 turning is located at the active datum
 - 3:** Do not change axis couplings
- ▶ **Q536 Eccentric turning without stop?:** 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

Cycles: Turning

12.3 RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801)

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

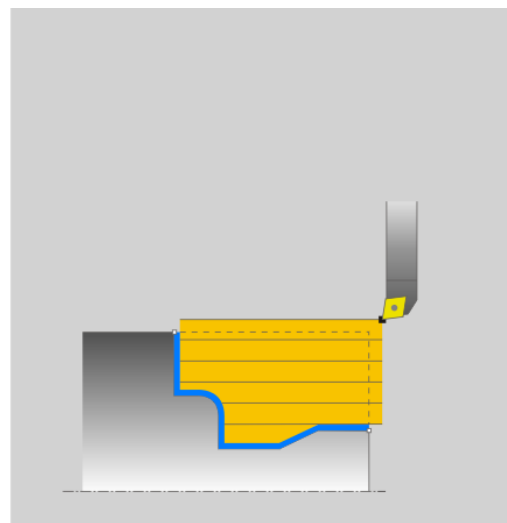
The pre-positioning of the tool decisively affects the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when a cycle is called. When calculating the area to be machined, the TNC takes into account the starting point and 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 339).

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.

Cycles: Turning

12.5 TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811)

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.

TURN SHOULDER LONGITUDINAL 12.5 (Cycle 811, DIN/ISO: G811)

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.

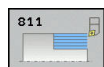
Please note while programming:



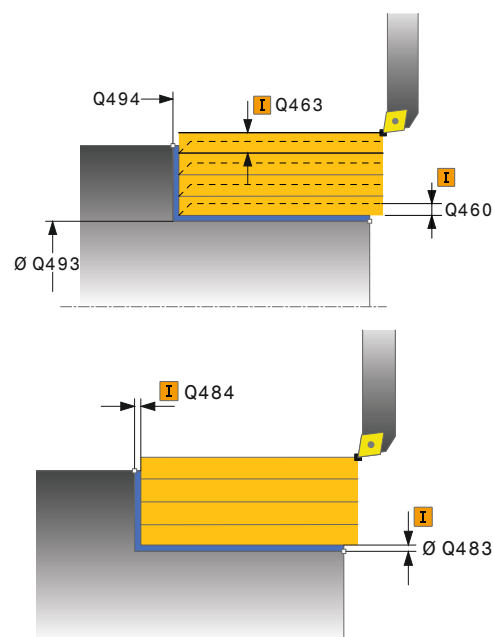
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
The tool position at cycle call defines the size of the area to be machined (cycle starting point).
Also refer to the fundamentals of turning cycles (see page 349).

12.5 TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q506 Contour smoothing (0/1/2)?**:
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 SHOULDER, LONGITDNL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
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	

TURN SHOULDER LONGITUDINAL EXTENDED 12.6 (Cycle 812, DIN/ISO: G812)

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.

Cycles: Turning

12.6 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812)

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 **RO** before the cycle call.

The tool position at cycle call (cycle starting point) affects the area to be machined.

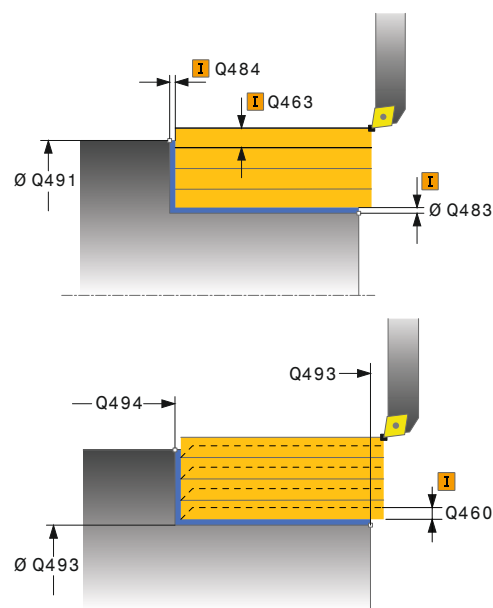
Also refer to the fundamentals of turning cycles (see page 349).

TURN SHOULDER LONGITUDINAL EXTENDED 12.6 (Cycle 812, DIN/ISO: G812)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of circumferen. surface?**: Angle between the circumferential surface and the rotary axis



12.6 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812)

- ▶ **Q501 Starting element type (0/1/2)?**: 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
- ▶ **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of face?**: Angle between the face and the rotary axis
- ▶ **Q503 End element type (0/1/2)?**: 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
- ▶ **Q504 Size of end element?**: Size of the end element (chamfer section)
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q506 Contour smoothing (0/1/2)?**:
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 SHOULDER, LONG. EXT.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
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	

TURN, LONGITUDINAL PLUNGE 12.7 (Cycle 813, DIN/ISO: G813)

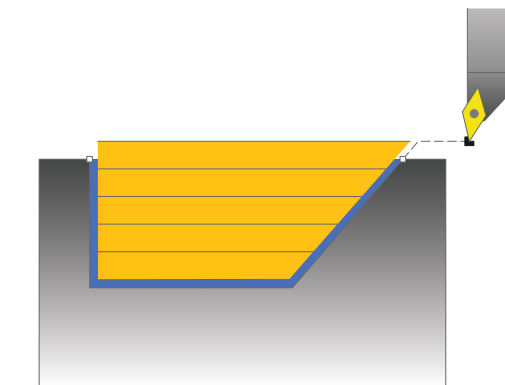
12.7 TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813)

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.

Cycles: Turning

12.7 TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813)

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 349).

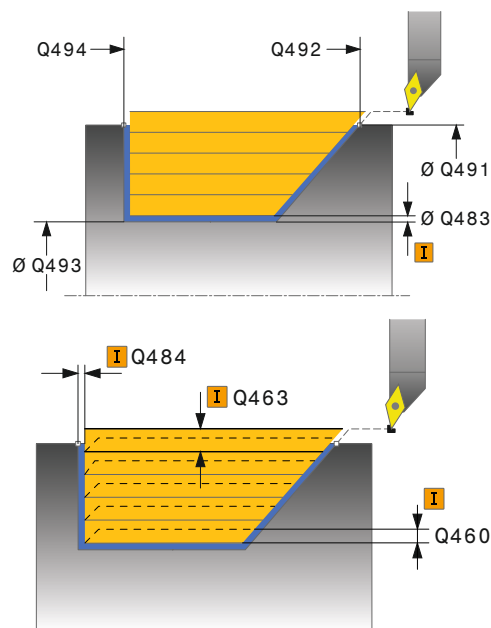
TURN, LONGITUDINAL PLUNGE 12.7

(Cycle 813, DIN/ISO: G813)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the perpendicular to the rotary axis.
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q506 Contour smoothing (0/1/2)?**:
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 PLUNGE CONTOUR LONGITUDINAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-10	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
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	

Cycles: Turning

12.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)

12.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)

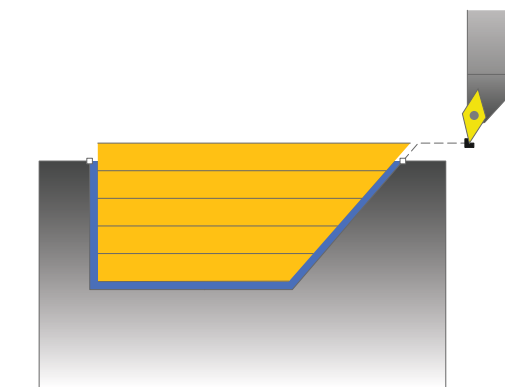
Application

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.

TURN, LONGITUDINAL PLUNGE EXTENDED 12.8 (Cycle 814, DIN/ISO: G814)

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.

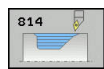
If complete machining with the active tool is not possible, a warning is output by the TNC.

Also refer to the fundamentals of turning cycles (see page 349).

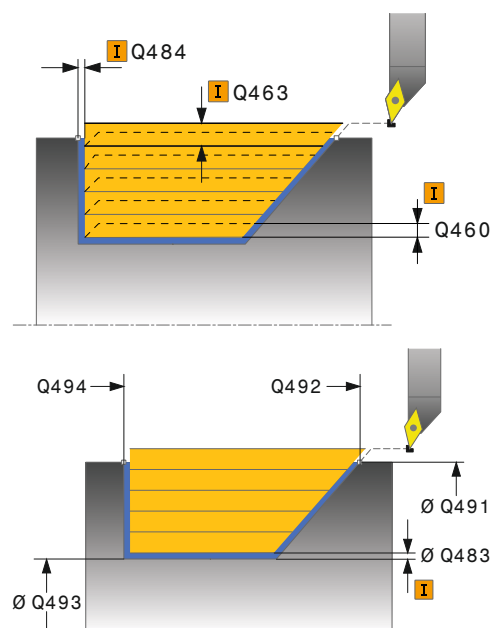
Cycles: Turning

12.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the perpendicular to the rotary axis.



TURN, LONGITUDINAL PLUNGE EXTENDED 12.8

(Cycle 814, DIN/ISO: G814)

- ▶ **Q501 Starting element type (0/1/2)?**: 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
- ▶ **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of face?**: Angle between the face and the rotary axis
- ▶ **Q503 End element type (0/1/2)?**: 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
- ▶ **Q504 Size of end element?**: Size of the end element (chamfer section)
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q506 Contour smoothing (0/1/2)?**:
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 PLUNGE LONGITUDINAL 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 CONTOUR END
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	

Cycles: Turning

12.9 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)

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.

TURN CONTOUR LONGITUDINAL 12.9 (Cycle 810, DIN/ISO: G810)

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 **RO** 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 349).

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

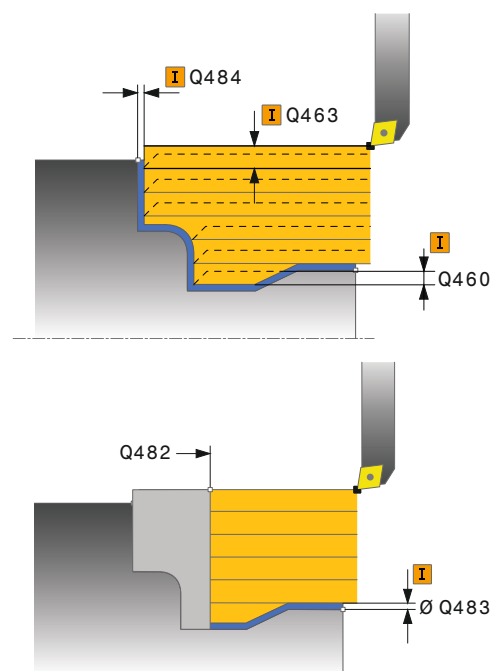
Cycles: Turning

12.9 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
0: Contour machined in the programmed direction
1: Contour machined in reverse direction to the programmed direction
2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999



TURN CONTOUR LONGITUDINAL 12.9 (Cycle 810, DIN/ISO: G810)

- ▶ **Q478 Roughing feed rate?:** 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q487 Allow plunging (0/1)?:** Permit machining of plunging elements:
0: Do not machine plunging elements
1: Machine plunging elements
- ▶ **Q488 Feed rate for plunging (0=auto)?:** Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- ▶ **Q479 Machining limits (0/1)? :** Activate cutting limit:
0: No cutting limit active
1: Cutting limit (**Q480/Q482**)
- ▶ **Q480 Value of diameter limit?:** X value for contour limitation (diameter value)
- ▶ **Q482 Value of cutting limit in Z?:** Z value for contour limitation
- ▶ **Q506 Contour smoothing (0/1/2)?:**
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 ;CONTOUR MACHINING LIMIT
Q480=+0 ;DIAMETER LIMIT VALUE
Q482=+0 ;LIMIT VALUE 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

Cycles: Turning

12.10 TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)

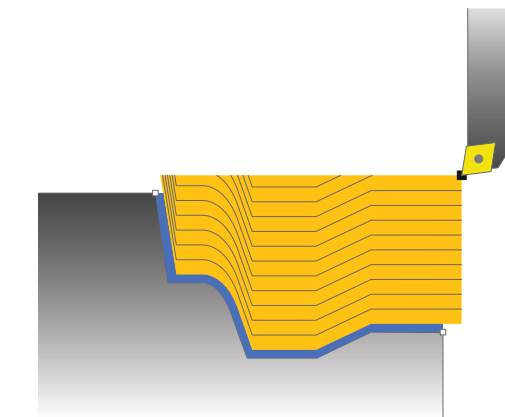
12.10 TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)

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.

TURN CONTOUR-PARALLEL 12.10 (Cycle 815, DIN/ISO: G815)

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 349).

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

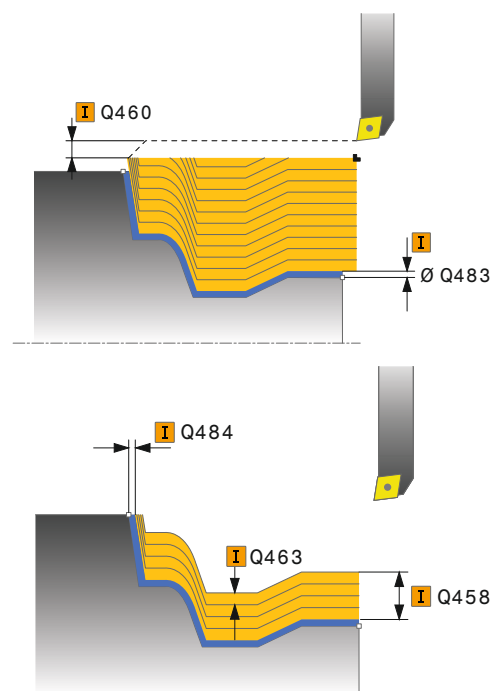
Cycles: Turning

12.10 TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q485 Allowance for workpiece blank?** (incremental): Contour-parallel oversize for the defined contour
- ▶ **Q486 Type of cut lines (=0/1)?**: Define the type of cutting lines:
0: Cuts with constant chip cross section
1: Equidistant proportioning of cuts
- ▶ **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
0: Contour machined in the programmed direction
1: Contour machined in reverse direction to the programmed direction
2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted



TURN CONTOUR-PARALLEL 12.10 (Cycle 815, DIN/ISO: G815)

- ▶ **Q463 Maximum cutting depth?:** Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q478 Roughing feed rate?:** 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?:** 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 CONTOUR-PAR. TURNING
Q215=+0 ;MACHINING OPERATION
Q460=+2 ;SAFETY CLEARANCE
Q485=+5 ;ALLOWANCE ON BLANK
Q486=+0 ;INTERSECTING LINES
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
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

Cycles: Turning

12.11 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)

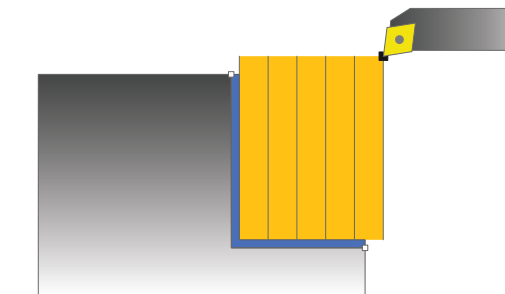
12.11 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)

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.

TURN SHOULDER FACE 12.11 (Cycle 821, DIN/ISO: G821)

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 349).

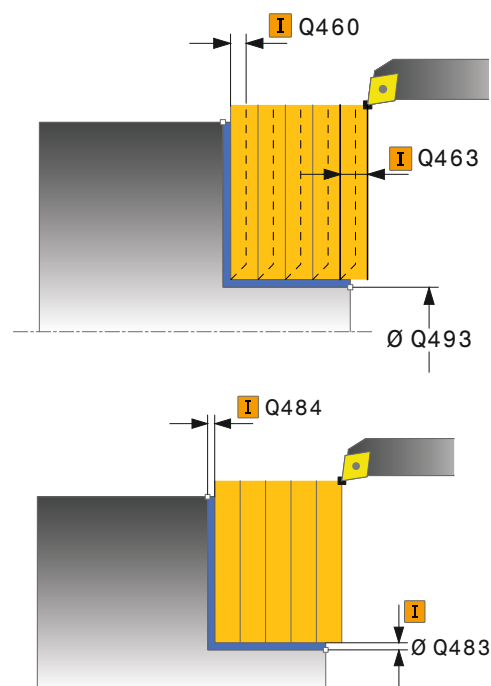
Cycles: Turning

12.11 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q506 Contour smoothing (0/1/2)?**:
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 SHOULDER, FACE	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+30	;DIAMETER AT CONTOUR END
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	

TURN SHOULDER FACE EXTENDED 12.12 (Cycle 822, DIN/ISO: G822)

12.12 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)

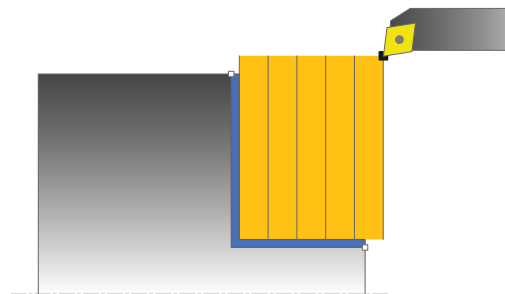
Application

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.

Cycles: Turning

12.12 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)

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.

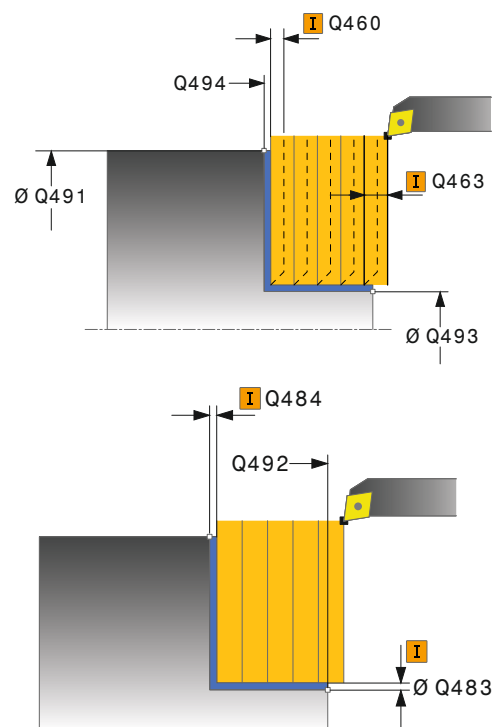
Also refer to the fundamentals of turning cycles (see page 349).

TURN SHOULDER FACE EXTENDED 12.12 (Cycle 822, DIN/ISO: G822)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of the face?**: Angle between the face and the rotary axis
- ▶ **Q501 Starting element type (0/1/2)?**: 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



12.12 TURN SHOULDER FACE EXTENDED

(Cycle 822, DIN/ISO: G822)

- ▶ **Q502 Size of starting element?:** Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?:** Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of circumferen. surface?:** Angle between the circumferential surface and the rotary axis
- ▶ **Q503 End element type (0/1/2)?:** 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
- ▶ **Q504 Size of end element?:** Size of the end element (chamfer section)
- ▶ **Q463 Maximum cutting depth?:** Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- ▶ **Q478 Roughing feed rate?:** 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q506 Contour smoothing (0/1/2)?:**
 - 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 SHOULDER, FACE. EXT.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+30	;DIAMETER AT CONTOUR END
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	

TURN, TRANSVERSE PLUNGE 12.13 (Cycle 823, DIN/ISO: G823)

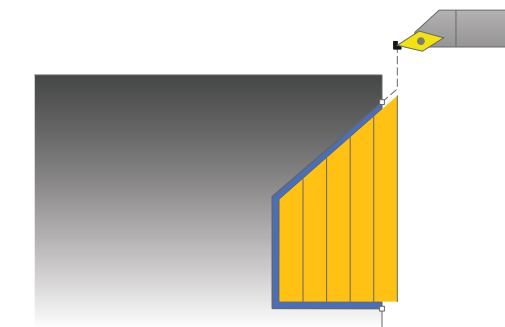
12.13 TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)

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.

Cycles: Turning

12.13 TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)

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.

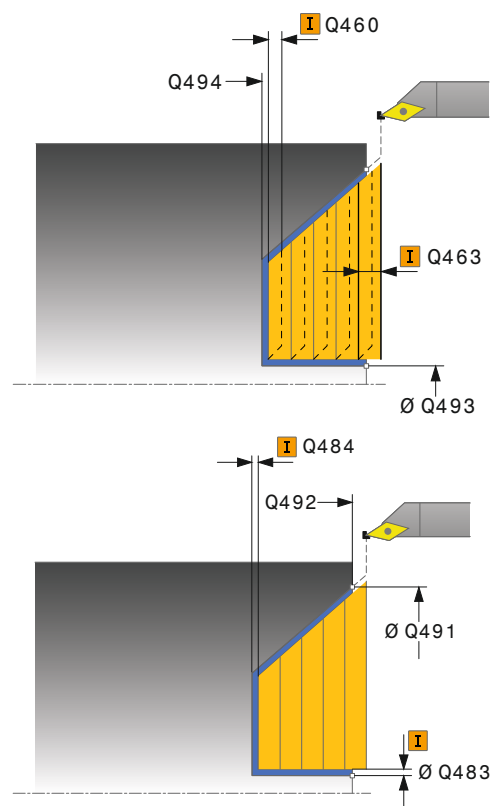
Also refer to the fundamentals of turning cycles (see page 349).

TURN, TRANSVERSE PLUNGE 12.13 (Cycle 823, DIN/ISO: G823)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?:** Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q491 Diameter at contour start?:** X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?:** Z coordinate of the starting point for the plunging path
- ▶ **Q493 Diameter at end of contour?:** X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?:** Z coordinate of the contour end point
- ▶ **Q495 Angle of side?:** Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- ▶ **Q463 Maximum cutting depth?:** Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- ▶ **Q478 Roughing feed rate?:** 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q506 Contour smoothing (0/1/2)?:**
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 CONTOUR END
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	

Cycles: Turning

12.14 TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)

12.14 TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)

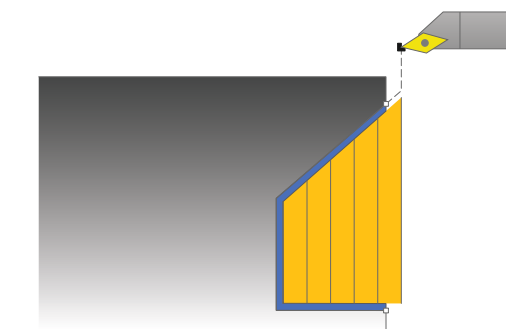
Application

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.

TURN, TRANSVERSE PLUNGE EXTENDED 12.14 (Cycle 824, DIN/ISO: G824)

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 349).

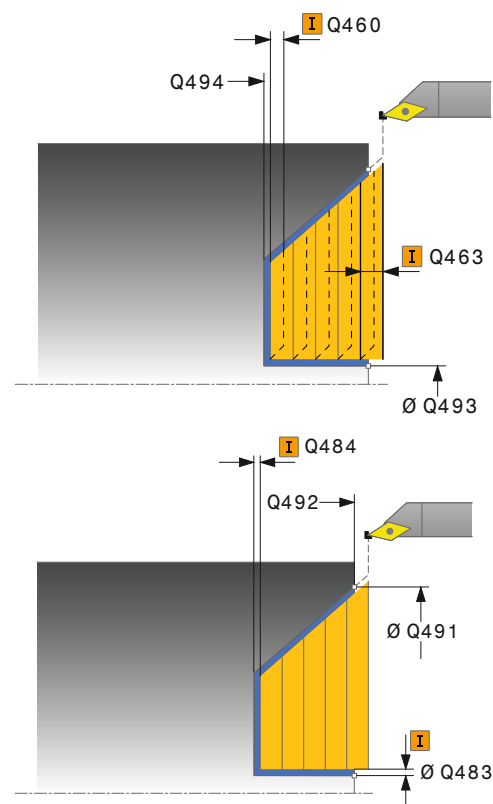
Cycles: Turning

12.14 TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q491 Diameter at contour start?**: X coordinate of the starting point for the plunging path (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- ▶ **Q501 Starting element type (0/1/2)?**: 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



TURN, TRANSVERSE PLUNGE EXTENDED 12.14 (Cycle 824, DIN/ISO: G824)

- ▶ **Q502 Size of starting element?:** Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?:** Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q503 End element type (0/1/2)?:** 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
- ▶ **Q504 Size of end element?:** Size of the end element (chamfer section)
- ▶ **Q463 Maximum cutting depth?:** Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- ▶ **Q478 Roughing feed rate?:** 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q506 Contour smoothing (0/1/2)?:**
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 PLUNGE TRANSVERSE 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 CONTOUR END
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	

Cycles: Turning

12.15 TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)

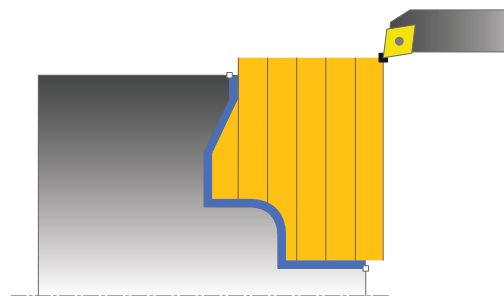
12.15 TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)

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.

TURN CONTOUR FACE 12.15 (Cycle 820, DIN/ISO: G820)

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 **RO** 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 349).

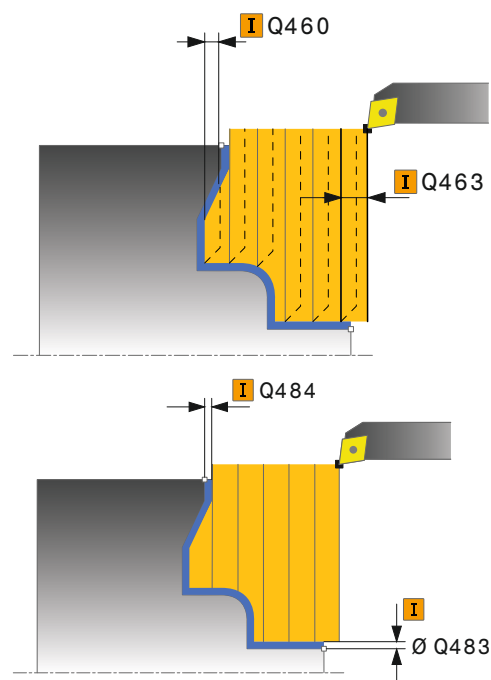
When you use local **QL Q** parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

12.15 TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- ▶ **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
0: Contour machined in the programmed direction
1: Contour machined in reverse direction to the programmed direction
2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- ▶ **Q478 Roughing feed rate?**: 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.



TURN CONTOUR FACE 12.15 (Cycle 820, DIN/ISO: G820)

- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q487 Allow plunging (0/1)?**: Permit machining of plunging elements:
0: Do not machine plunging elements
1: Machine plunging elements
- ▶ **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- ▶ **Q479 Machining limits (0/1)?**: Activate cutting limit:
0: No cutting limit active
1: Cutting limit (**Q480/Q482**)
- ▶ **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- ▶ **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- ▶ **Q506 Contour smoothing (0/1/2)?**:
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 TRANSV.
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 ;CONTOUR MACHINING LIMIT
Q480=+0 ;DIAMETER LIMIT VALUE
Q482=+0 ;LIMIT VALUE 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

Cycles: Turning

12.16 SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)

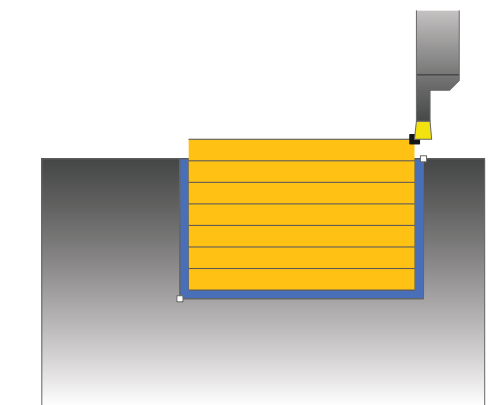
12.16 SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)

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.

SIMPLE RADIAL RECESSING 12.16 (Cycle 841, DIN/ISO: G841)

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).

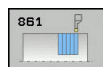
From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single 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).

Cycles: Turning

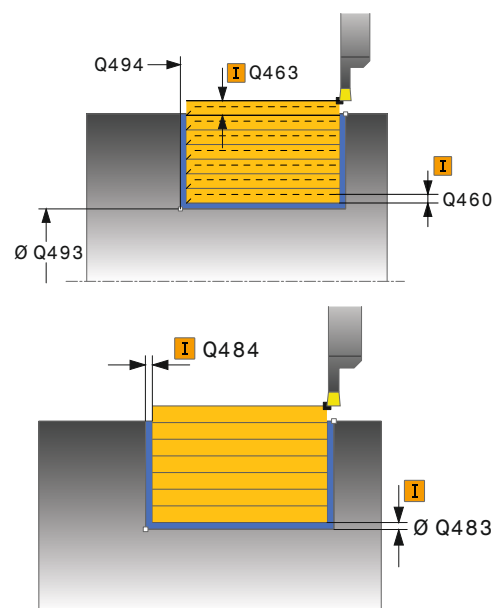
12.16 SIMPLE RADIAL RECESSING

(Cycle 841, DIN/ISO: G841)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in contour direction)
- ▶ **Q508 Offset width?**: 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.
- ▶ **Q509 Depth compensat. for finishing?**: 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.
- ▶ **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.



NC blocks

11 CYCL DEF 841 SIMPLE REC.
TURN., RADIAL DIR.

Q215=+0 ;MACHINING
OPERATION

Q460=+2 ;SAFETY CLEARANCE

Q493=+50 ;DIAMETER AT
CONTOUR END

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

RADIAL RECESSING EXTENDED 12.17 (Cycle 842, DIN/ISO: G842)

12.17 RADIAL RECESSING EXTENDED (Cycle 842, DIN/ISO: G842)

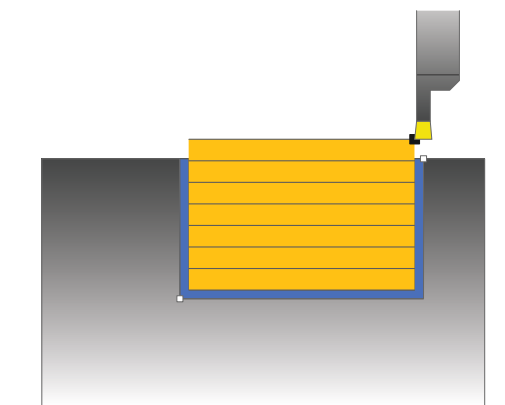
Application

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.

Cycles: Turning

12.17 RADIAL RECESSING EXTENDED

(Cycle 842, DIN/ISO: G842)

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).

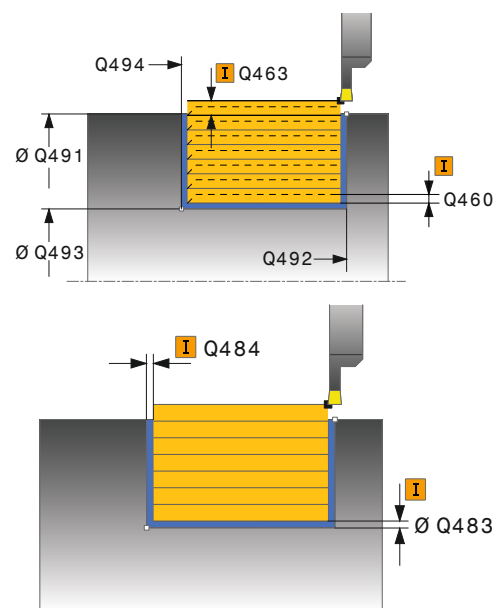
From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single 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).

RADIAL RECESSING EXTENDED 12.17 (Cycle 842, DIN/ISO: G842)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of side?**: Angle between the side at the contour starting point and the perpendicular to the rotary axis



12.17 RADIAL RECESSING EXTENDED

(Cycle 842, DIN/ISO: G842)

- ▶ **Q501 Starting element type (0/1/2)?**: 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
- ▶ **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of second side?**: Angle between the side at the contour end point and the perpendicular to the rotary axis
- ▶ **Q503 End element type (0/1/2)?**: 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
- ▶ **Q504 Size of end element?**: Size of the end element (chamfer section)
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in contour direction)
- ▶ **Q508 Offset width?**: 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 EXPND. RECESS, RADL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
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	

RADIAL RECESSING EXTENDED 12.17

(Cycle 842, DIN/ISO: G842)

- ▶ **Q509 Depth compensat. for finishing?:** 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.
- ▶ **Q488 Feed rate for plunging (0=auto)?:** Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

Cycles: Turning

12.18 RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)

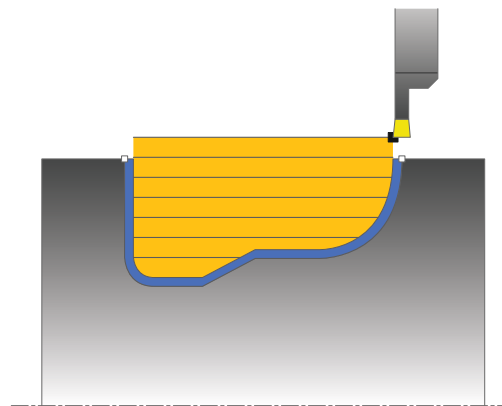
12.18 RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)

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.

RECESSING CONTOUR RADIAL 12.18 (Cycle 840, DIN/ISO: G840)

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).

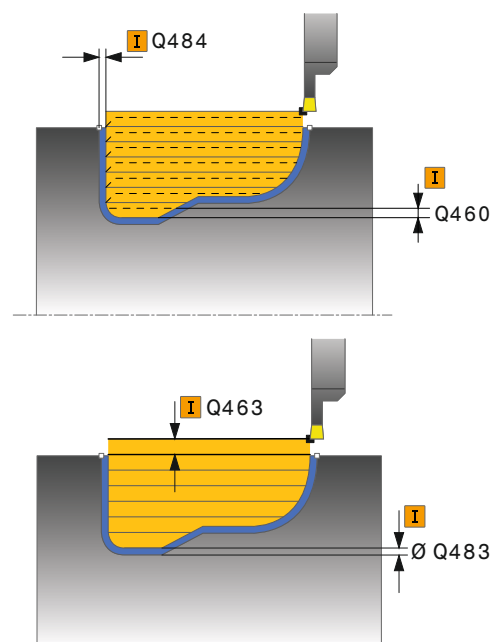
Cycles: Turning

12.18 RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction



RECESSING CONTOUR RADIAL 12.18 (Cycle 840, DIN/ISO: G840)

- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q479 Machining limits (0/1)? :** Activate cutting limit:
0: No cutting limit active
1: Cutting limit (**Q480/Q482**)
- ▶ **Q480 Value of diameter limit?:** X value for contour limitation (diameter value)
- ▶ **Q482 Value of cutting limit in Z?:** Z value for contour limitation
- ▶ **Q463 Maximum cutting depth?:** Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q507 Direction (0=bidir./1=unidir.)?:** Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in contour direction)
- ▶ **Q508 Offset width?:** 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.
- ▶ **Q509 Depth compensat. for finishing?:** 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.
- ▶ **Q499 Reverse contour (0=no/1=yes)?:** 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, RADIAL
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 ;CONTOUR MACHINING LIMIT
Q480=+0 ;DIAMETER LIMIT VALUE
Q482=+0 ;LIMIT VALUE 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

Cycles: Turning

12.19 SIMPLE AXIAL RECESSING

(Cycle 851, DIN/ISO: G851)

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.

SIMPLE AXIAL RECESSING 12.19 (Cycle 851, DIN/ISO: G851)

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).

Cycles: Turning

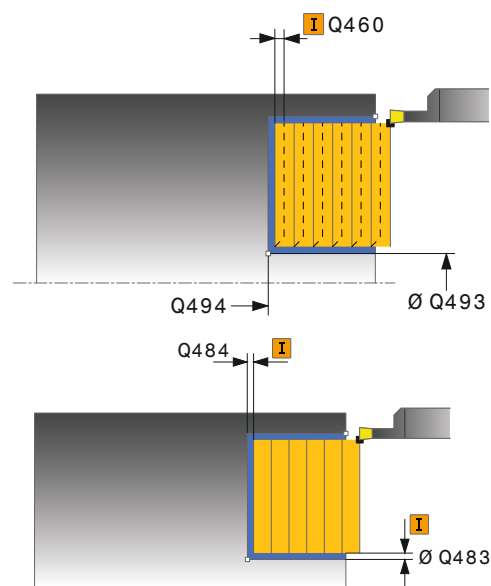
12.19 SIMPLE AXIAL RECESSING

(Cycle 851, DIN/ISO: G851)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in contour direction)
- ▶ **Q508 Offset width?**: 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.
- ▶ **Q509 Depth compensat. for finishing?**: 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.
- ▶ **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.



NC blocks

11 CYCL DEF 851 SIMPLE REC TURNG, AX

Q215=+0 ;MACHINING OPERATION

Q460=+2 ;SAFETY CLEARANCE

Q493=+50 ;DIAMETER AT CONTOUR END

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

AXIAL RECESSING EXTENDED 12.20 (Cycle 852, DIN/ISO: G852)

12.20 AXIAL RECESSING EXTENDED (Cycle 852, DIN/ISO: G852)

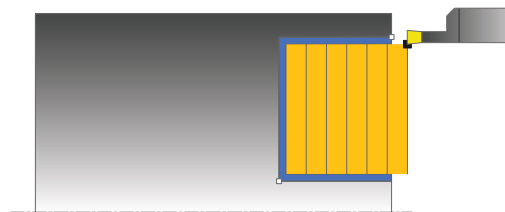
Application

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.

Cycles: Turning

12.20 AXIAL RECESSING EXTENDED

(Cycle 852, DIN/ISO: G852)

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:



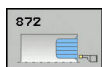
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The tool position at cycle call defines the size of the area to be machined (cycle starting point).

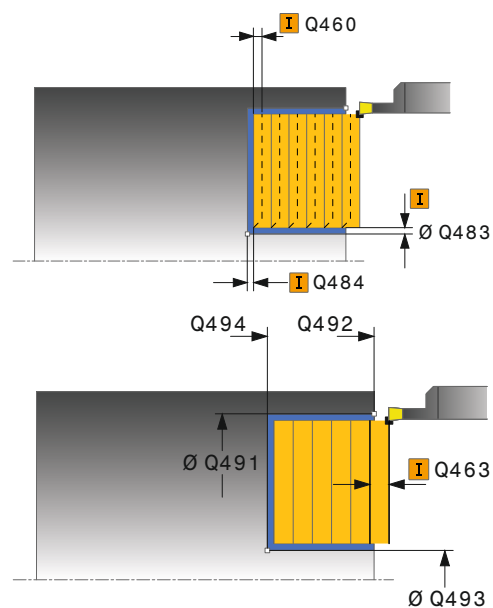
From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single 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).

AXIAL RECESSING EXTENDED 12.20 (Cycle 852, DIN/ISO: G852)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?:** Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?:** Reserved, currently without function
- ▶ **Q491 Diameter at contour start?:** X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?:** Z coordinate of the contour starting point
- ▶ **Q493 Diameter at end of contour?:** X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?:** Z coordinate of the contour end point
- ▶ **Q495 Angle of side?:** Angle between the side at the contour starting point and the parallel line to the rotary axis



12.20 AXIAL RECESSING EXTENDED

(Cycle 852, DIN/ISO: G852)

- ▶ **Q501 Starting element type (0/1/2)?**: 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
- ▶ **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of second side?**: Angle between the side at the contour end point and the parallel line to the rotary axis
- ▶ **Q503 End element type (0/1/2)?**: 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
- ▶ **Q504 Size of end element?**: Size of the end element (chamfer section)
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in contour direction)
- ▶ **Q508 Offset width?**: 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 ENH.REC.TURNING, AX.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
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	

AXIAL RECESSING EXTENDED 12.20 (Cycle 852, DIN/ISO: G852)

- ▶ **Q509 Depth compensat. for finishing?:** 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.
- ▶ **Q488 Feed rate for plunging (0=auto)?:** Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

Cycles: Turning

12.21 AXIAL RECESSING

(Cycle 850, DIN/ISO: G850)

12.21 AXIAL RECESSING

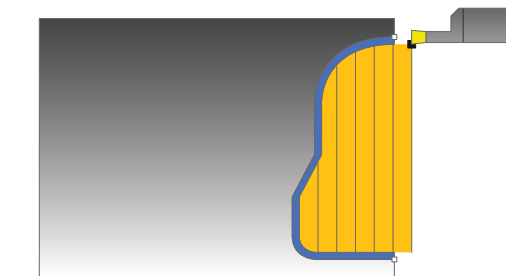
(Cycle 850, DIN/ISO: G850)

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.

AXIAL RECESSING 12.21 (Cycle 850, DIN/ISO: G850)

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.

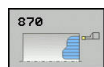
From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single 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).

Cycles: Turning

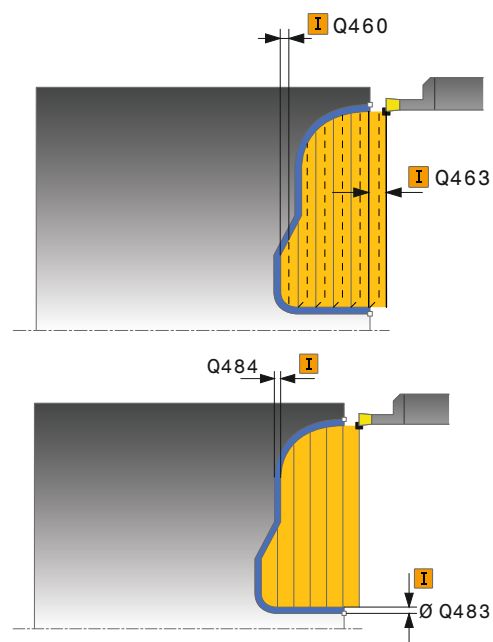
12.21 AXIAL RECESSING

(Cycle 850, DIN/ISO: G850)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction



AXIAL RECESSING 12.21

(Cycle 850, DIN/ISO: G850)

- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q479 Machining limits (0/1)? :** Activate cutting limit:
0: No cutting limit active
1: Cutting limit (**Q480/Q482**)
- ▶ **Q480 Value of diameter limit?:** X value for contour limitation (diameter value)
- ▶ **Q482 Value of cutting limit in Z?:** Z value for contour limitation
- ▶ **Q463 Maximum cutting depth?:** Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q507 Direction (0=bidir./1=unidir.)?:** Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in contour direction)
- ▶ **Q508 Offset width?:** 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.
- ▶ **Q509 Depth compensat. for finishing?:** 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.
- ▶ **Q499 Reverse contour (0=no/1=yes)?:** 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
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 ;CONTOUR MACHINING LIMIT
Q480=+0 ;DIAMETER LIMIT VALUE
Q482=+0 ;LIMIT VALUE 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

Cycles: Turning

12.22 RADIAL RECESSING

(Cycle 861, DIN/ISO: G861)

12.22 RADIAL RECESSING

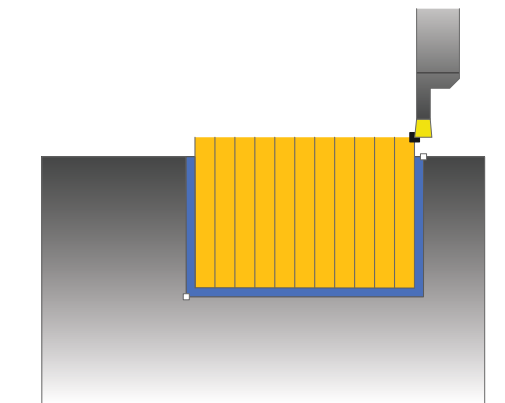
(Cycle 861, DIN/ISO: G861)

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.

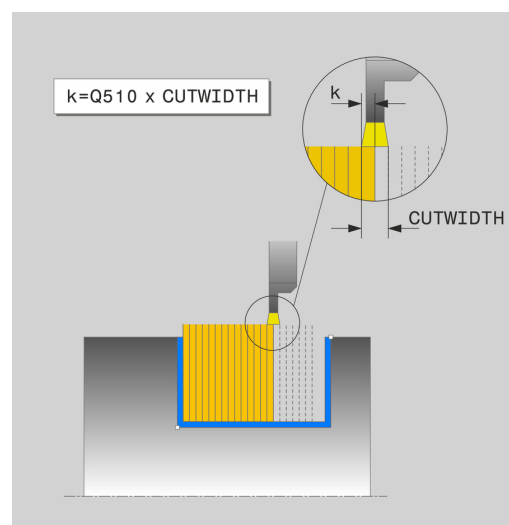
The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle processes only the area from the cycle starting point to the end point defined in the cycle.

- 1 For the first full recess, the TNC moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
- 2 The TNC retracts the tool at rapid traverse.
- 3 The TNC performs a stepover of the value **Q510** x tool width (**CUTWIDTH**).
- 4 The TNC then recesses again, this time with the feed rate **Q478**
- 5 The TNC retracts the tool as per parameter **Q462**
- 6 The TNC cuts the area between the starting position and the end point by repeating steps 2 through 4
- 7 As soon as the slot width has been achieved, the TNC positions the tool at rapid traverse back to the cycle starting point.



RADIAL RECESSING 12.22 (Cycle 861, DIN/ISO: G861)

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.

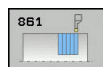
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: $CUTWIDTH + DCW_{Tab} + FUNCTION\ TURNDATA\ CORR\ TCS: Z/X\ DCW$. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

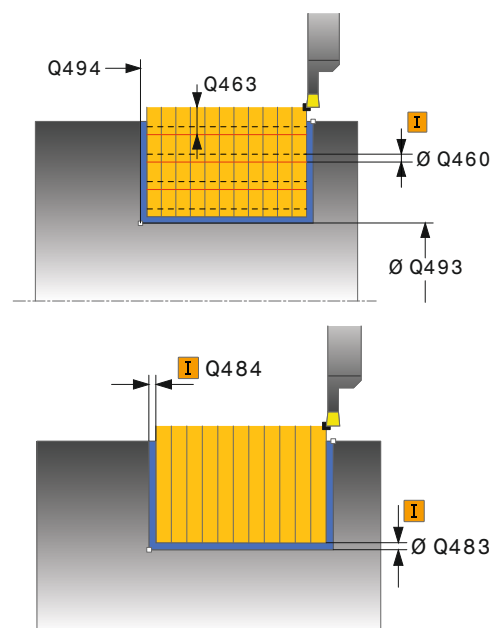
12.22 RADIAL RECESSING

(Cycle 861, DIN/ISO: G861)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Limit to plunging depth?**: Max. recess depth per cut
- ▶ **Q510 Overlap factor for recess width?** The factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the width **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range 0.001 to 1



NC blocks

11 CYCL DEF 861 SIMPLE RECESS, RADL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE

RADIAL RECESSING 12.22 (Cycle 861, DIN/ISO: G861)

- ▶ **Q511 Feed rate factor in %2** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The TNC thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total this can lead to reduced machining times. Input range 0.001 to 150
- ▶ **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
0: The TNC retracts the tool along the contour
1: The TNC first moves the tool away from the contour diagonally and then retracts it

Q483=+0.4 ;OVERSIZE FOR
DIAMETER

Q484=+0.2 ;OVERSIZE IN Z

Q505=+0.2 ;FINISHING FEED RATE

Q463=+0 ;LIMIT TO DEPTH

Q510=+0.8 ;RECESSING OVERLAP

Q511=+100;FEED RATE FACTOR

Q462=0 ;RETRACTION MODE

12 L X+75 Y+0 Z-25 FMAX M303

13 CYCL CALL

Cycles: Turning

12.23 RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)

12.23 RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)

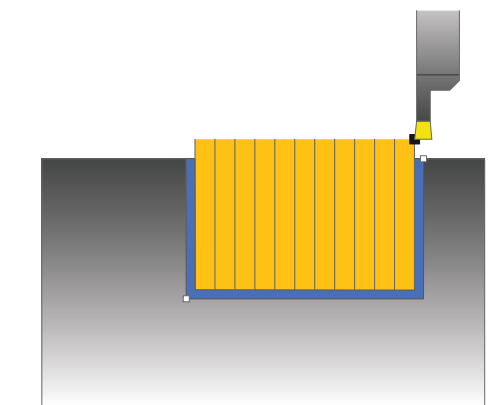
Application

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

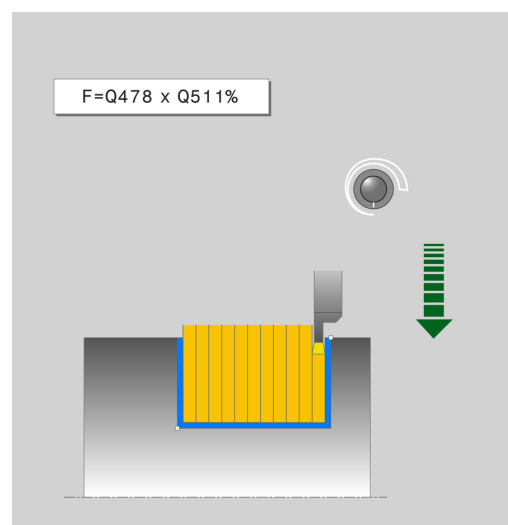
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

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 For the first full recess, the TNC moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
- 2 The TNC retracts the tool at rapid traverse.
- 3 The TNC performs a stepover of the value **Q510** x tool width (**CUTWIDTH**).
- 4 The TNC then recesses again, this time with the feed rate **Q478**
- 5 The TNC retracts the tool as per parameter **Q462**
- 6 The TNC cuts the area between the starting position and the end point by repeating steps 2 through 4
- 7 As soon as the slot width has been achieved, the TNC positions the tool at rapid traverse back to the cycle starting point.



RADIAL RECESSING EXTENDED 12.23 (Cycle 862, DIN/ISO: G862)

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).

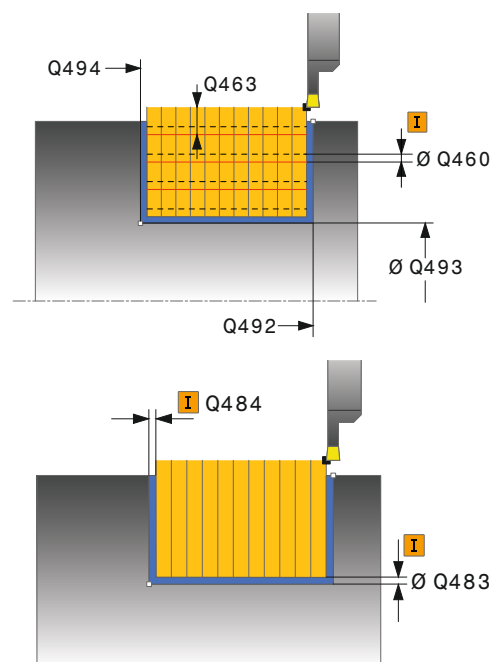
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: $CUTWIDTH + DCW_{Tab} + FUNCTION\ TURNDATA\ CORR\ TCS: Z/X\ DCW$. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

12.23 RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q495 Angle of side?**: Angle between the side at the contour starting point and the perpendicular to the rotary axis
- ▶ **Q501 Starting element type (0/1/2)?**: 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
- ▶ **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of second side?**: Angle between the side at the contour end point and the perpendicular to the rotary axis
- ▶ **Q503 End element type (0/1/2)?**: 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 862 EXPND. RECESS,
RADL.

Q215=+0 ;MACHINING
OPERATION

Q460=+2 ;SAFETY CLEARANCE

Q491=+75 ;DIAMETER AT
CONTOUR START

Q492=-20 ;CONTOUR START IN Z

Q493=+50 ;DIAMETER AT
CONTOUR END

Q494=-50 ;CONTOUR END IN Z

Q495=+5 ;ANGLE OF SIDE

Q501=+1 ;TYPE OF STARTING
ELEMENT

RADIAL RECESSING EXTENDED 12.23

(Cycle 862, DIN/ISO: G862)

- ▶ **Q504 Size of end element?:** Size of the end element (chamfer section)
- ▶ **Q478 Roughing feed rate?:** 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q463 Limit to plunging depth?:** Max. recess depth per cut
- ▶ **Q510 Overlap factor for recess width?** The factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the width **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range 0.001 to 1
- ▶ **Q511 Feed rate factor in %2** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The TNC thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total this can lead to reduced machining times. Input range 0.001 to 150
- ▶ **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
0: The TNC retracts the tool along the contour
1: The TNC first moves the tool away from the contour diagonally and then retracts it

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

Q510=0.8 ;RECESSING OVERLAP

Q511=+100;FEED RATE FACTOR

Q462=+0 ;RETRACTION MODE

12 L X+75 Y+0 Z+2 FMAX M303

13 CYCL CALL

Cycles: Turning

12.24 RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)

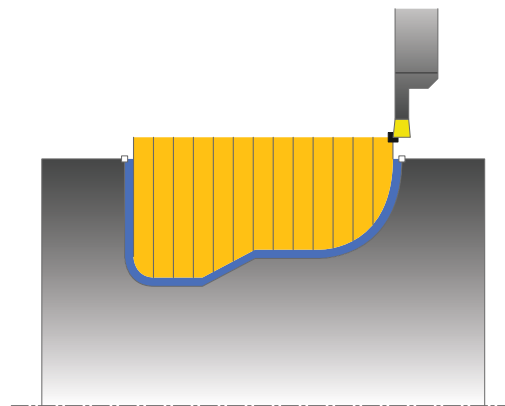
12.24 RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)

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 For the first full recess, the TNC moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
- 2 The TNC retracts the tool at rapid traverse.
- 3 The TNC performs a stepover of the value **Q510** x tool width (**CUTWIDTH**).
- 4 The TNC then recesses again, this time with the feed rate **Q478**
- 5 The TNC retracts the tool as per parameter **Q462**
- 6 The TNC cuts the area between the starting position and the end point by repeating steps 2 through 4
- 7 As soon as the slot width has been achieved, the TNC positions the tool at rapid traverse back to the cycle starting point.

RECESSING CONTOUR RADIAL 12.24 (Cycle 860, DIN/ISO: G860)

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.

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: $CUTWIDTH + DCW_{Tab} + FUNCTION\ TURNDATA\ CORR\ TCS: Z/X\ DCW$. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

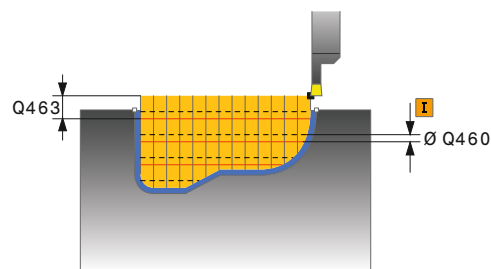
Cycles: Turning

12.24 RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)

Cycle parameters



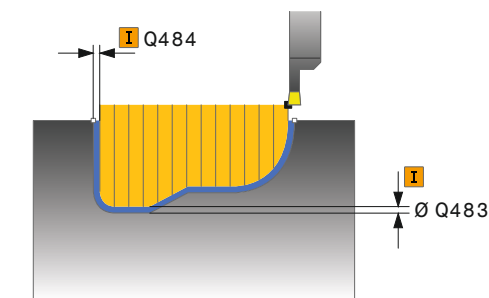
- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- ▶ **Q483 Oversize for diameter?** (incremental):
Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction



RECESSING CONTOUR RADIAL 12.24

(Cycle 860, DIN/ISO: G860)

- ▶ **Q505 Finishing feed rate?:** 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.
- ▶ **Q479 Machining limits (0/1)? :** Activate cutting limit:
0: No cutting limit active
1: Cutting limit (**Q480/Q482**)
- ▶ **Q480 Value of diameter limit?:** X value for contour limitation (diameter value)
- ▶ **Q482 Value of cutting limit in Z?:** Z value for contour limitation
- ▶ **Q463 Limit to plunging depth?:** Max. recess depth per cut
- ▶ **Q510 Overlap factor for recess width?** The factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the width **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range 0.001 to 1
- ▶ **Q511 Feed rate factor in %2** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The TNC thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total this can lead to reduced machining times. Input range 0.001 to 150
- ▶ **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
0: The TNC retracts the tool along the contour
1: The TNC first moves the tool away from the contour diagonally and then retracts it



NC blocks

9	CYCL DEF 14.0 CONTOUR
10	CYCL DEF 14.1 CONTOUR LABEL2
11	CYCL DEF 860 CONT. RECESS, 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	;CONTOUR MACHINING LIMIT
Q480=+0	;DIAMETER LIMIT VALUE
Q482=+0	;LIMIT VALUE Z
Q463=+0	;LIMIT TO DEPTH
Q510=0.08	;RECESSING OVERLAP
Q511=+100	;FEED RATE FACTOR
Q462=+0	;RETRACTION MODE
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

Cycles: Turning

12.25 AXIAL RECESSING

(Cycle 871, DIN/ISO: G871)

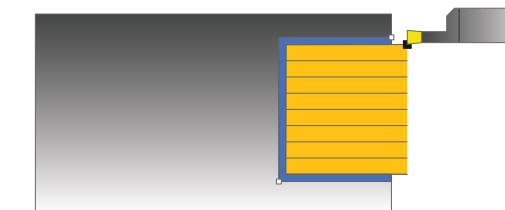
12.25 AXIAL RECESSING

(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 For the first full recess, the TNC moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
- 2 The TNC retracts the tool at rapid traverse.
- 3 The TNC performs a stepover of the value **Q510** x tool width (**CUTWIDTH**).
- 4 The TNC then recesses again, this time with the feed rate **Q478**
- 5 The TNC retracts the tool as per parameter **Q462**
- 6 The TNC cuts the area between the starting position and the end point by repeating steps 2 through 4
- 7 As soon as the slot width has been achieved, the TNC positions the tool at rapid traverse back 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.

AXIAL RECESSING 12.25 (Cycle 871, DIN/ISO: G871)

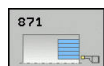
Please note while programming:



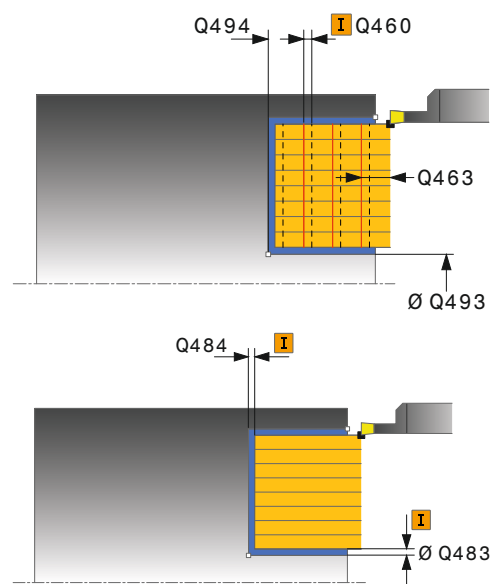
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: $CUTWIDTH + DCW_{Tab} + FUNCTION\ TURNDATA\ CORR\ TCS: Z/X\ DCW$. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
 - 0**: Roughing and finishing
 - 1**: Only roughing
 - 2**: Only finishing to finished dimension
 - 3**: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the contour end point
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Limit to plunging depth?**: Max. recess depth per cut
- ▶ **Q510 Overlap factor for recess width?** The factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the width **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range 0.001 to 1



NC blocks

11 CYCL DEF 871 SIMPLE RECESS, AXIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-10	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE

12.25 AXIAL RECESSING

(Cycle 871, DIN/ISO: G871)

- ▶ **Q511 Feed rate factor in %2** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The TNC thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total this can lead to reduced machining times. Input range 0.001 to 150
- ▶ **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
0: The TNC retracts the tool along the contour
1: The TNC first moves the tool away from the contour diagonally and then retracts it

Q483=+0.4 ;OVERSIZE FOR
DIAMETER

Q484=+0.2 ;OVERSIZE IN Z

Q505=+0.2 ;FINISHING FEED RATE

Q463=+0 ;LIMIT TO DEPTH

Q510=+0.8 ;RECESSING OVERLAP

Q511=+100;FEED RATE FACTOR

Q462=0 ;RETRACTION MODE

12 L X+65 Y+0 Z+2 FMAX M303

13 CYCL CALL

AXIAL RECESSING EXTENDED 12.26 (Cycle 872, DIN/ISO: G872)

12.26 AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)

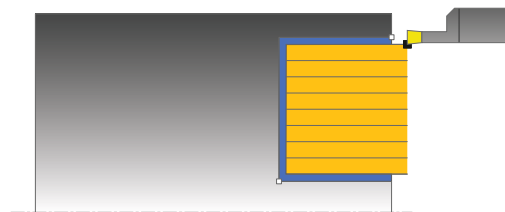
Application

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 For the first full recess, the TNC moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
- 2 The TNC retracts the tool at rapid traverse.
- 3 The TNC performs a stepover of the value **Q510** x tool width (**CUTWIDTH**).
- 4 The TNC then recesses again, this time with the feed rate **Q478**
- 5 The TNC retracts the tool as per parameter **Q462**
- 6 The TNC cuts the area between the starting position and the end point by repeating steps 2 through 4
- 7 As soon as the slot width has been achieved, the TNC positions the tool at rapid traverse back to the cycle starting point.

Cycles: Turning

12.26 AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)

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).

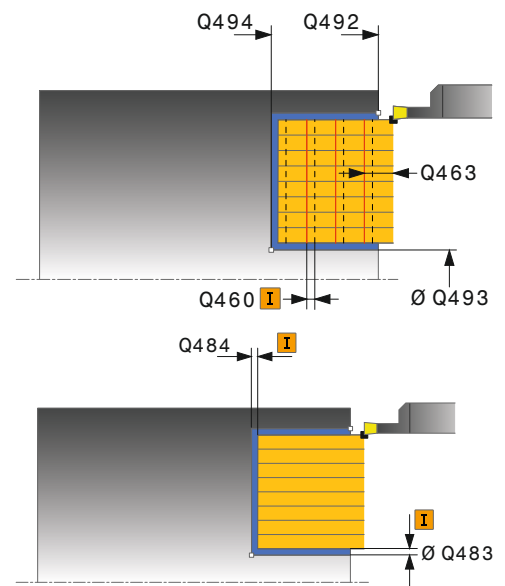
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: $CUTWIDTH + DCW_{Tab} + FUNCTION\ TURNDATA\ CORR\ TCS: Z/X\ DCW$. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

AXIAL RECESSING EXTENDED 12.26 (Cycle 872, DIN/ISO: G872)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?:** Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?:** Reserved, currently without function
- ▶ **Q491 Diameter at contour start?:** X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?:** Z coordinate of the contour starting point
- ▶ **Q493 Diameter at end of contour?:** X coordinate of the contour end point (diameter value)
- ▶ **Q494 Contour end in Z?:** Z coordinate of the contour end point
- ▶ **Q495 Angle of side?:** Angle between the side at the contour starting point and the parallel line to the rotary axis



12.26 AXIAL RECESSING EXTENDED

(Cycle 872, DIN/ISO: G872)

- ▶ **Q501 Starting element type (0/1/2)?**: 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
- ▶ **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- ▶ **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius of the cutting insert is generated.
- ▶ **Q496 Angle of second side?**: Angle between the side at the contour end point and the parallel line to the rotary axis
- ▶ **Q503 End element type (0/1/2)?**: 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
- ▶ **Q504 Size of end element?**: Size of the end element (chamfer section)
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.
- ▶ **Q463 Limit to plunging depth?**: Max. recess depth per cut
- ▶ **Q510 Overlap factor for recess width?** The factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the width **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range 0.001 to 1

NC blocks

11 CYCL DEF 871 EXPND. RECESS, AXIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
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
Q510=+0.08	RECESSING OVERLAP
Q511=+100	;FEED RATE FACTOR
Q462=0	;RETRACTION MODE
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

AXIAL RECESSING EXTENDED 12.26 (Cycle 872, DIN/ISO: G872)

- ▶ **Q511 Feed rate factor in %2** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The TNC thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total this can lead to reduced machining times. Input range 0.001 to 150
- ▶ **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
 - 0:** The TNC retracts the tool along the contour
 - 1:** The TNC first moves the tool away from the contour diagonally and then retracts it

Cycles: Turning

12.27 AXIAL RECESSING

(Cycle 870, DIN/ISO: G870)

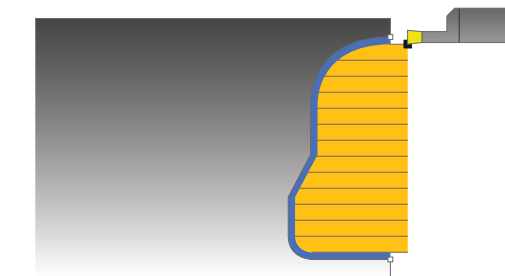
12.27 AXIAL RECESSING

(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 For the first full recess, the TNC moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
- 2 The TNC retracts the tool at rapid traverse.
- 3 The TNC performs a stepover of the value **Q510** x tool width (**CUTWIDTH**).
- 4 The TNC then recesses again, this time with the feed rate **Q478**
- 5 The TNC retracts the tool as per parameter **Q462**
- 6 The TNC cuts the area between the starting position and the end point by repeating steps 2 through 4
- 7 As soon as the slot width has been achieved, the TNC positions the tool at rapid traverse back to the cycle starting point.

AXIAL RECESSING 12.27 (Cycle 870, DIN/ISO: G870)

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.

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: $CUTWIDTH + DCW_{Tab} + FUNCTION\ TURNDATA\ CORR\ TCS: Z/X\ DCW$. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

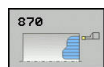
When you use local **QL Q** parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

Cycles: Turning

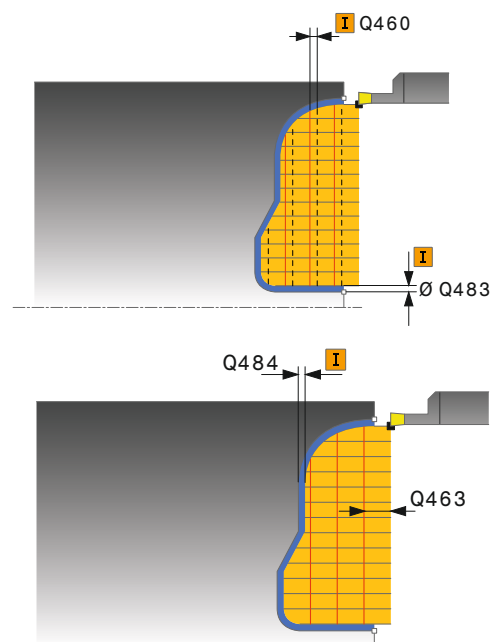
12.27 AXIAL RECESSING

(Cycle 870, DIN/ISO: G870)

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q460 Set-up clearance?**: Reserved, currently without function
- ▶ **Q478 Roughing feed rate?**: 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.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour
- ▶ **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- ▶ **Q505 Finishing feed rate?**: 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.



AXIAL RECESSING 12.27

(Cycle 870, DIN/ISO: G870)

- ▶ **Q479 Machining limits (0/1)?** : Activate cutting limit:
0: No cutting limit active
1: Cutting limit (**Q480/Q482**)
- ▶ **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- ▶ **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- ▶ **Q463 Limit to plunging depth?**: Max. recess depth per cut
- ▶ **Q510 Overlap factor for recess width?** The factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the width **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range 0.001 to 1
- ▶ **Q511 Feed rate factor in %2** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The TNC thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total this can lead to reduced machining times. Input range 0.001 to 150
- ▶ **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
0: The TNC retracts the tool along the contour
1: The TNC first moves the tool away from the contour diagonally and then retracts it

NC blocks

9	CYCL DEF 14.0 CONTOUR
10	CYCL DEF 14.1 CONTOUR LABEL2
11	CYCL DEF 870 CONT. RECESS, 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	;CONTOUR MACHINING LIMIT
Q480=+0	;DIAMETER LIMIT VALUE
Q482=+0	;LIMIT VALUE Z
Q463=+0	;LIMIT TO DEPTH
Q510=0.8	;RECESSING OVERLAP
Q511=+100	;FEED RATE FACTOR
Q462=+0	;RETRACTION MODE
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

Cycles: Turning

12.28 THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)

12.28 THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)

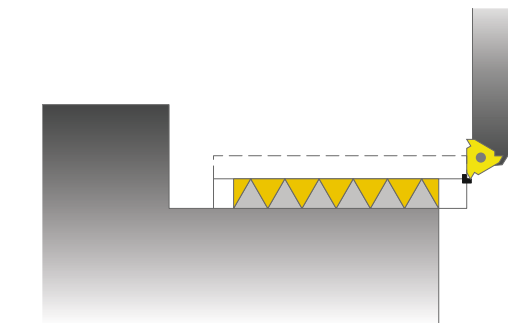
Application

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.

THREAD LONGITUDINAL 12.28 (Cycle 831, DIN/ISO: G831)

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).



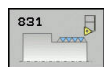
With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use 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.

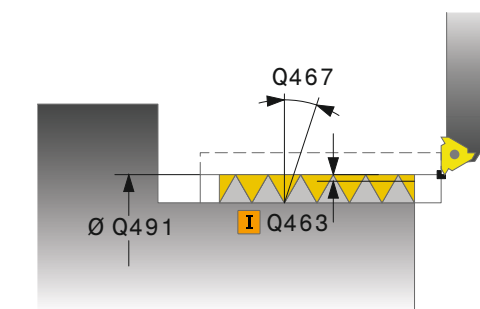
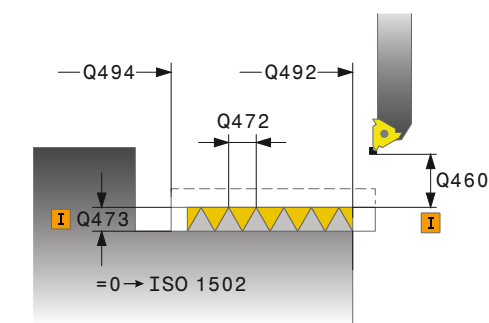
Cycles: Turning

12.28 THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)

Cycle parameters



- ▶ **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
0: External thread
1: Internal thread
- ▶ **Q460 Setup clearance?**: 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.
- ▶ **Q491 Thread diameter?**: Define the nominal diameter of the thread.
- ▶ **Q472 Thread pitch?**: Pitch of the thread
- ▶ **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- ▶ **Q492 Contour start in Z?**: Z coordinate of the starting point
- ▶ **Q494 Contour end in Z?**: Z coordinate of the end point including the runout of the thread Q474.
- ▶ **Q474 Length of thread runout?** (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.
- ▶ **Q463 Maximum cutting depth?**: Maximum plunging depth in radial direction relative to the radius.
- ▶ **Q467 Feed angle?**: Angle for the infeed Q463. The reference angle is formed by the perpendicular to the rotary axis.
- ▶ **Q468 Infeed type (0/1)?**: Define the type of infeed:
0: Constant chip cross section (infeed lessens with depth)
1: Constant plunging depth
- ▶ **Q470 Starting angle?**: Angle of the turning spindle at which the thread start is to be made.
- ▶ **Q475 Number of thread grooves?**: Number of thread grooves
- ▶ **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth



NC blocks

11 CYCL DEF 831 THREAD LONGITUDINAL
Q471=+0 ;THREAD POSITION
Q460=+5 ;SAFETY CLEARANCE
Q491=+75 ;THREAD DIAMETER
Q472=+2 ;THREAD PITCH
Q473=+0 ;DEPTH OF THREAD
Q492=+0 ;CONTOUR START IN Z
Q494=-15 ;CONTOUR END IN Z
Q474=+0 ;THREAD RUN-OUT
Q463=+0.5 ;MAX. CUTTING DEPTH
Q467=+30 ;ANGLE OF INFEEED
Q468=+0 ;TYPE OF INFEEED
Q470=+0 ;STARTING ANGLE
Q475=+30 ;NUMBER OF STARTS
Q476=+30 ;NUMBER 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

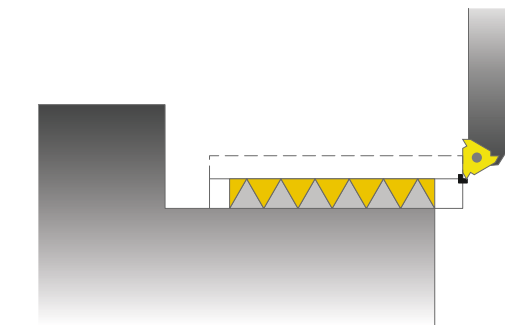
This cycle enables you to run both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- 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.

12.29 THREAD EXTENDED (Cycle 832, DIN/ISO: G832)

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).

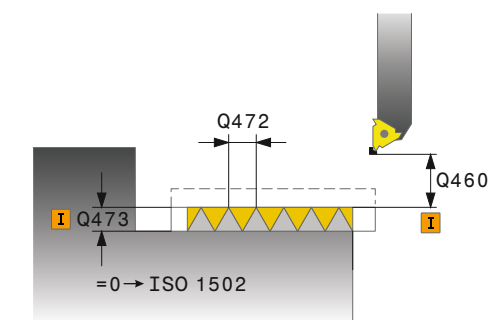
The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.

THREAD EXTENDED (Cycle 832, DIN/ISO: G832) 12.29

Cycle parameters



- ▶ **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
0: External thread
1: Internal thread
- ▶ **Q461 Thread orientation (0/1)?**: Define the direction of the thread pitch:
0: Longitudinal (parallel to the rotary axis)
1: Lateral (perpendicular to the rotary axis)
- ▶ **Q460 Set-up clearance?**: Set-up clearance perpendicular to thread pitch.
- ▶ **Q472 Thread pitch?**: Pitch of the thread
- ▶ **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- ▶ **Q464 Dimension type taper (0-4)?**: 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
- ▶ **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- ▶ **Q492 Contour start in Z?**: Z coordinate of the starting point
- ▶ **Q493 Diameter at end of contour?**: X coordinate of the end point (diameter value)
- ▶ **Q494 Contour end in Z?**: Z coordinate of the end point
- ▶ **Q469 Taper angle (diameter)?**: Taper angle of contour
- ▶ **Q474 Length of thread runout?** (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.
- ▶ **Q465 Starting path?** (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.
- ▶ **Q466 Overrun path?**: Length of the path in pitch direction on which the feed axes are decelerated. The overrun path is within the defined thread contour.
- ▶ **Q463 Maximum cutting depth?**: Maximum plunging depth perpendicular to the thread pitch
- ▶ **Q467 Feed angle?**: Angle for the infeed Q463. The reference angle is formed by the parallel line to the thread pitch.



NC blocks

11 CYCL DEF 832 THREAD EXTENDED	
Q471=+0	;THREAD POSITION
Q461=+0	;THREAD ORIENTATION
Q460=+2	;SAFETY CLEARANCE
Q472=+2	;THREAD PITCH
Q473=+0	;DEPTH OF THREAD
Q464=+0	;DIMENSION TYPE TAPER
Q491=+100	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+110	;DIAMETER AT CONTOUR END
Q494=-35	;CONTOUR END IN Z
Q469=+0	;TAPER ANGLE
Q474=+0	;THREAD RUN-OUT
Q465=+4	;STARTING PATH
Q466=+4	;OVERRUN PATH
Q463=+0.5	;MAX. CUTTING DEPTH
Q467=+30	;ANGLE OF INFEEED
Q468=+0	;TYPE OF INFEEED
Q470=+0	;STARTING ANGLE
Q475=+30	;NUMBER OF STARTS
Q476=+30	;NUMBER 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)

- ▶ **Q468 Infeed type (0/1)?**: Define the type of infeed:
0: Constant chip cross section (infeed lessens with depth)
1: Constant plunging depth
- ▶ **Q470 Starting angle?**: Angle of the turning spindle at which the thread start is to be made.
- ▶ **Q475 Number of thread grooves?**: Number of thread grooves
- ▶ **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth

CONTOUR-PARALLEL THREAD 12.30 (Cycle 830, DIN/ISO: G830)

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.

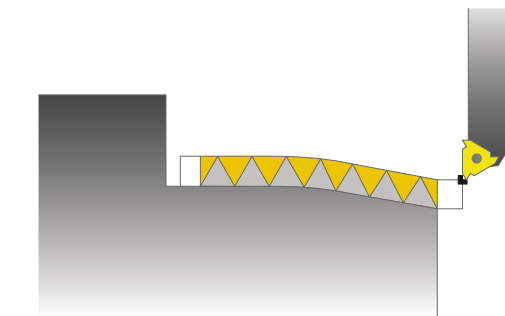
You can process single threads or multi-threads with the cycle.

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.

Cycles: Turning

12.30 CONTOUR-PARALLEL THREAD

(Cycle 830, DIN/ISO: G830)

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).

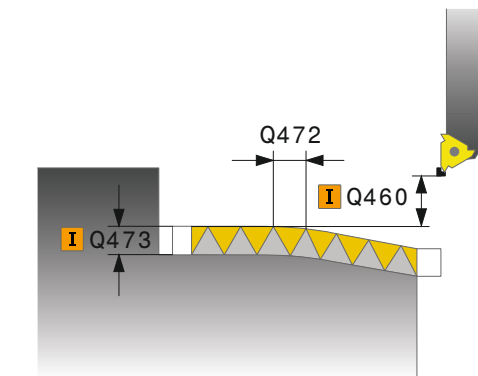
The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.

CONTOUR-PARALLEL THREAD 12.30 (Cycle 830, DIN/ISO: G830)

Cycle parameters



- ▶ **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
0: External thread
1: Internal thread
- ▶ **Q461 Thread orientation (0/1)?**: Define the direction of the thread pitch:
0: Longitudinal (parallel to the rotary axis)
1: Lateral (perpendicular to the rotary axis)
- ▶ **Q460 Set-up clearance?**: Set-up clearance perpendicular to thread pitch.
- ▶ **Q472 Thread pitch?**: Pitch of the thread
- ▶ **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- ▶ **Q474 Length of thread runout?** (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.

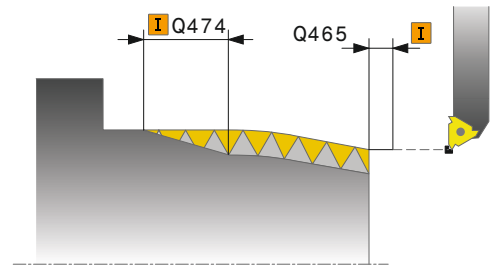


12

Cycles: Turning

12.30 CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ISO: G830)

- ▶ **Q465 Starting path?** (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.
- ▶ **Q466 Overrun path?**: Length of the path in pitch direction on which the feed axes are decelerated. The overrun path is within the defined thread contour.
- ▶ **Q463 Maximum cutting depth?**: Maximum plunging depth perpendicular to the thread pitch
- ▶ **Q467 Feed angle?**: Angle for the infeed Q463. The reference angle is formed by the parallel line to the thread pitch.
- ▶ **Q468 Infeed type (0/1)?**: Define the type of infeed:
0: Constant chip cross section (infeed lessens with depth)
1: Constant plunging depth
- ▶ **Q470 Starting angle?**: Angle of the turning spindle at which the thread start is to be made.
- ▶ **Q475 Number of thread grooves?**: Number of thread grooves
- ▶ **Q476 Number of air cuts?**: 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 THREAD CONTOUR-PARALLEL
	Q471=+0 ;THREAD POSITION
	Q461=+0 ;THREAD ORIENTATION
	Q460=+2 ;SAFETY CLEARANCE
	Q472=+2 ;THREAD PITCH
	Q473=+0 ;DEPTH OF THREAD
	Q474=+0 ;THREAD RUN-OUT
	Q465=+4 ;STARTING PATH
	Q466=+4 ;OVERRUN PATH
	Q463=+0.5 ;MAX. CUTTING DEPTH
	Q467=+30 ;ANGLE OF INFEEED
	Q468=+0 ;TYPE OF INFEEED
	Q470=+0 ;STARTING ANGLE
	Q475=+30 ;NUMBER OF STARTS
	Q476=+30 ;NUMBER 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)

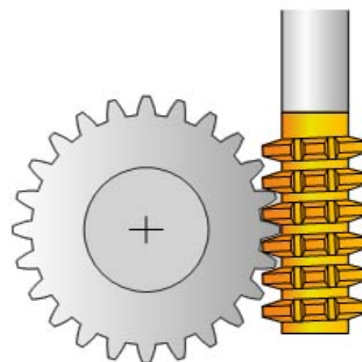
Cycle run

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**.



Cycles: Turning

12.31 GEAR HOBGING (Cycle 880, DIN/ISO: G880)

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: $Q541 \times S$. 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

Software option 131 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 HOBGING remember to call Cycle 801 and M145 in order to reset the coordinate system.

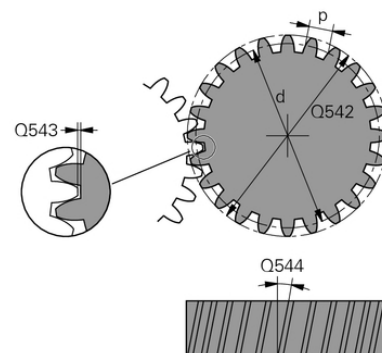
If you abort a program during machining, you must **reset the coordinate system with Cycle 801** and **call M145** before you start machining again!

GEAR HOBBING (Cycle 880, DIN/ISO: G880) 12.31

Cycle parameters



- ▶ **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing to finished dimension
3: Only finishing to oversize
- ▶ **Q540 Module?**: Define the gear: Module of the gear wheel. Input range 0 to 99.9999
- ▶ **Q541 Number of teeth?**: Define the gear: Number of teeth. Input range 0 to 99999
- ▶ **Q542 Outside diameter?**: Define the gear: Outside diameter of the finished part. Input range 0 to 99999.9999
- ▶ **Q543 Trough-to-tip clearance?**: 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
- ▶ **Q544 Angle of inclination?**: Define the gear: Angle by which helical teeth are inclined relative to the direction of the axis (For straight-cut gears this angle is 0°.) Input range -45 to +45
- ▶ **Q545 Tool lead angle?**: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range : -60.0000 to +60.0000
- ▶ **Q546 Reverse tool rotation direction?**: Define the tool: Direction of spindle rotation of the gear hob:
3: Tool turns to the right (M3)
4: Tool turns to the left (M4)
- ▶ **Q547 Angle offset of tool spindle?**: Angle by which the TNC rotates the workpiece at the beginning of the cycle. Input range -180.0000 to +180.0000
- ▶ **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
0: Positive machining side
1: Negative machining side
- ▶ **Q533 Preferred dir. of incid. angle?**: Selection of alternate possibilities of inclination.
0: Option with shortest way
-1: Option with negative direction
+1: Option with positive direction
-2: Option with negative direction at a range of between -90° and -180°
+2: Option with positive direction at a range of between +90° and +180°



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	;SAFETY 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

12.31 GEAR HOBGING (Cycle 880, DIN/ISO: G880)

- ▶ **Q530 Inclined machining?:** Position the tilting axes for inclined machining:
 - 1:** Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between the workpiece and tool remains unchanged. The TNC performs a compensating movement with the linear axes
 - 2:** Position the tilting axis automatically without orienting the tool tip (TURN)
- ▶ **Q253 Feed rate for pre-positioning?:** 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**
- ▶ **Q553 TOOL:L offset, machining start?:** 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
- ▶ **Q551 Starting point in Z?:** Starting point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- ▶ **Q552 End point in Z?:** End point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- ▶ **Q463 Maximum cutting depth?:** Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- ▶ **Q460 Set-up clearance? (incremental):** Distance for retraction and pre-positioning Input range 0 to 999.999
- ▶ **Q488 Feed rate for plunging:** Feed rate for tool infeed. Input range 0 to 99999.999

GEAR HOBBING (Cycle 880, DIN/ISO: G880) 12.31

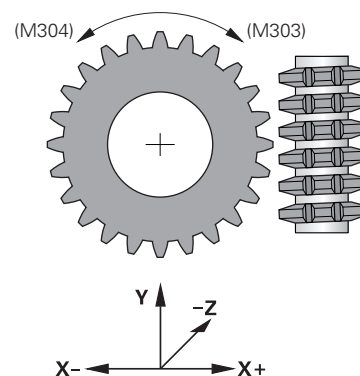
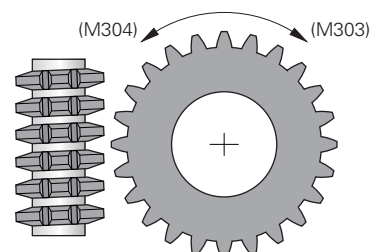
- ▶ **Q478 Roughing feed rate?:** Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- ▶ **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour .
- ▶ **Q505 Finishing feed rate?:** 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)



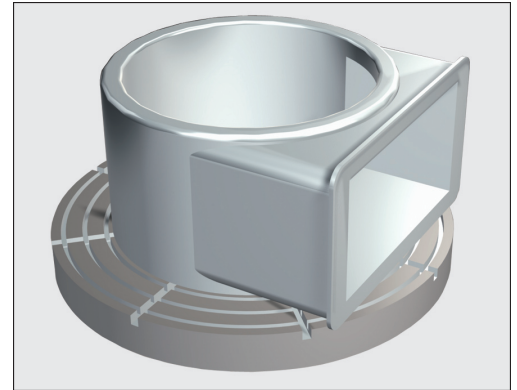
Cycles: Turning

12.32 CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)

12.32 CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)

Application

An unbalance can occur when turning a nonsymmetrical workpiece, such as a pump body. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle **892 CHECK IMBALANCE** 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.** If the maximum unbalance is not exceeded, the TNC executes the program without interruption. This function protects the machine mechanics. It enables you to take action if excessive unbalance is detected.



CHECK UNBALANCE (Cycle 892, DIN/ISO: G892) 12.32

Please note while programming:



Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance.

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.

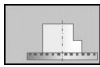
You need to check the applicability of the control's internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.



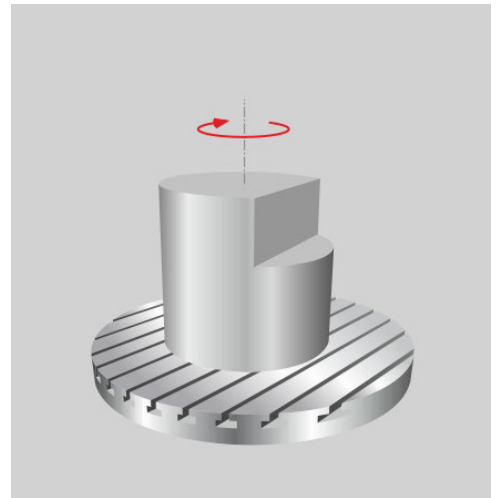
If Cycle 892 CHECK UNBALANCE has aborted a program, it is recommended to use the manual MEASURE UNBALANCE cycle. With this cycle, the TNC determines the unbalance and calculates the mass and position of a balancing weight. For more information on the manual MEASURE UNBALANCE cycle, refer to the User's Manual for Conversational Programming.

12.32 CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)

Cycle parameters



- ▶ **Q450 Max. permissible runout?** (mm): Indicates the maximum runout of a sinusoidal unbalance signal. The signal results from the following error of the measuring axis and from the spindle revolutions.
- ▶ **Q451 Rotational speed?** (rpm): The test for an unbalance begins with a low initial speed (e.g. 50 rpm). It is then automatically increased by specified increments (e.g. 25 rpm) until the defined maximum speed is reached. Spindle speed override is disabled.



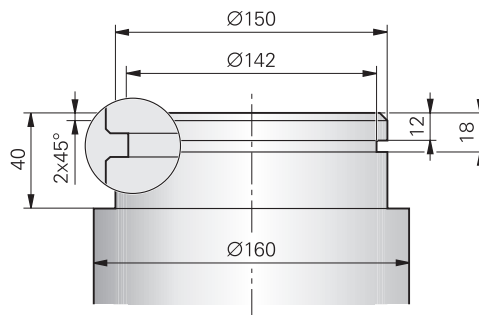
NC blocks

63 CYCL DEF 892 CHECK IMBALANCE	
Q450=0	;MAXIMUM RUNOUT
Q451=50	;SPEED

Example program 12.33

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 ADJUST XZ SYSTEM	Cycle definition adapt rotary coordinate system
Q497=+0 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=0 ;INCLINED MACHINING	
Q531=+0 ;ANGLE OF INCIDENCE?	
Q532=750 ;FEED RATE	
Q533=+0 ;PREFERRED DIRECTION	
Q535=3 ;ECCENTRIC TURNING	
Q536=0 ;ECCENTRIC W/O STOP	
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. EXT.	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 CONTOUR END	
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	

Cycles: Turning

12.33 Example program

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 ADJUST XZ SYSTEM		Cycle definition adapt rotary coordinate system
Q497=+0	;PRECESSION ANGLE	
Q498=+0	;REVERSE TOOL	
Q530=0	;INCLINED MACHINING	
Q531=+0	;ANGLE OF INCIDENCE?	
Q532=750	;FEED RATE	
Q533=+0	;PREFERRED DIRECTION	
Q535=0	;ECCENTRIC TURNING	
Q536=+0	;ECCENTRIC W/O STOP	
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 EXPND. RECESS, RADL.		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 CONTOUR END	
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	

Example program 12.33

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 run

- 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
5 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM		Reset the coordinate system
6 M145		Deactivate M144 if still active
7 FUNCTION TURNDATA SPIN VCONST:OFF S50		Constant surface speed OFF
8 M140 MB MAX		Retract the tool
9 L A+0 R0 FMAX		Set the rotary axis to 0
10 L X+250 Y-250 R0 FMAX		Pre-position the tool in the working plane on the side on which machining will be performed
11 Z+20 R0 FMAX		Pre-position the tool in the spindle axis
12 L M136		Feed rate in mm/rev
13 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	

12.33 Example program

Q546=+3	;CHANGE TOOL DIRECTN.	
Q547=+0	;ANG. OFFSET, SPINDLE	
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	
Q460=2	;SAFETY CLEARANCE	
Q488=+1	;PLUNGING FEED RATE	
Q478=+2	;ROUGHING FEED RATE	
Q483=+0.4	;OVERSIZE FOR DIAMETER	
Q505=+1	;FINISHING FEED RATE	
14 CYCL CALL M303		Call the cycle, spindle on
15 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM		Reset the coordinate system
16 M145		Deactivate the active M144 in the cycle
17 FUNCTION MODE MILL		Activate milling mode
18 M140 MB MAX		Retract the tool in the tool axis
19 L A+0 C+0 R0 FMAX		Reset the rotation
20 M30		END of program
21 END PGM 5 MM		

13

**Using Touch Probe
Cycles**

Using Touch Probe Cycles

13.1 General information about 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

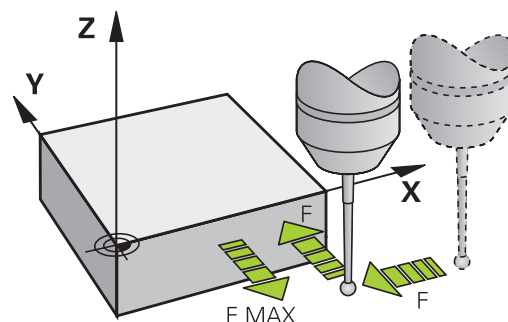
Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further Information: "Before You Start Working with Touch Probe Cycles", page 465

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 **Electronic handwheel** modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensating workpiece misalignment
- Setting datums

General information about touch probe cycles 13.1

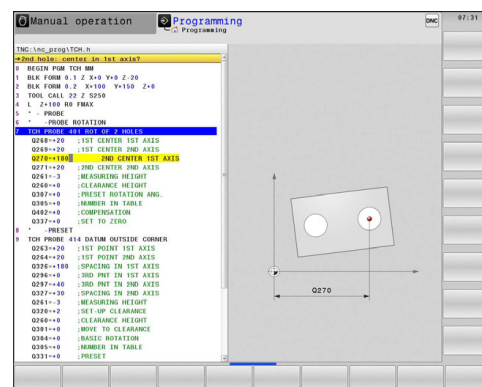
Touch probe cycles for automatic operation

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** 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).



Using Touch Probe Cycles

13.1 General information about touch probe cycles

Defining the touch probe cycle in the Programming mode of operation



- ▶ 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

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 ;NUMBER IN TABLE

Q331=+0 ;PRESET

Q332=+0 ;PRESET

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 ;PRESET

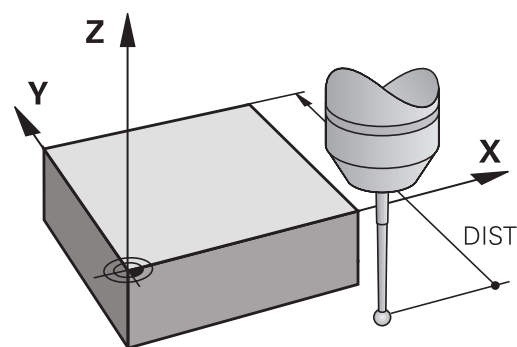
Soft key	Group of measuring cycles	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	472
	Cycles for automatic workpiece presetting	492
	Cycles for automatic workpiece inspection	546
	Special cycles	592
	Calibrate TS	592
	Kinematics	639
	Cycles for automatic tool measurement (enabled by the machine tool builder)	670
	Visual Setup Control (Option 136 VSC)	616

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.

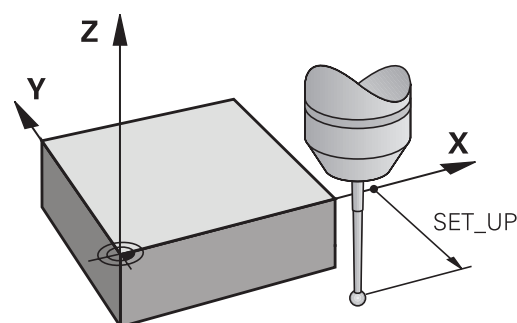
Maximum traverse to touch point: **DIST** in touch probe table

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.



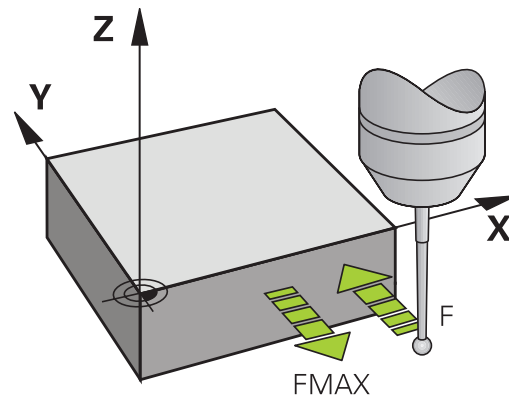
If you change **TRACK = ON**, you must recalibrate the touch probe.

Using Touch Probe Cycles

13.2 Before You Start Working with Touch Probe Cycles

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

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 SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR** 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

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 use 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:



- ▶ Operating mode: Press the **Manual operation** key



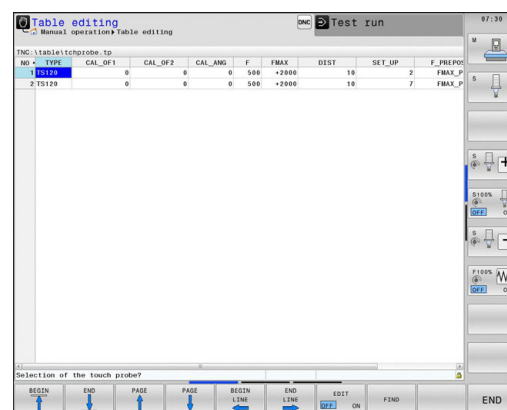
- ▶ To choose the touch probe functions, press the **TOUCH PROBE** soft key. The TNC displays additional soft keys



- ▶ To 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.
- ▶ To leave 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 the touch probe?
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	TS center misalignmt. ref. axis? [mm]
CAL_OF2	Offset of the touch probe axis to the spindle axis in the minor axis	TS center misalignmt. aux. axis? [mm]
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	Spindle angle for calibration?
F	Feed rate at which the control will probe the workpiece	Probing feed rate? [mm/min]
FMAX	Feed rate at which the touch probe is pre-positioning and is positioned between the measuring points	Rapid traverse in probing cycle? [mm/min]
DIST	If the stylus is not coordinated within this defined value, the control will issue an error message.	Maximum measuring range? [mm]
SET_UP	In SET_UP you define how far from the defined or calculated touch point the control 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: <ul style="list-style-type: none"> ■ Pre-positioning with speed from FMAX: FMAX_PROBE ■ Pre-positioning with machine rapid traverse: FMAX_MACHINE 	Pre-position at rapid? ENT/NOENT
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: <ul style="list-style-type: none"> ■ ON: Perform spindle tracking ■ OFF: Do not perform spindle tracking 	Probe oriented? Yes=ENT/No=NOENT

14

**Touch Probe
Cycles: Automatic
Measurement
of Workpiece
Misalignment**

14.1 Fundamentals

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.

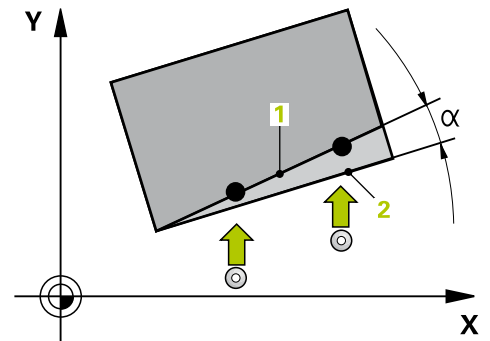
Refer to your machine manual.

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:

Soft key	Cycle	Page
	400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.	474
	401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.	477
	402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.	480
	403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.	483
	405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.	487
	404 SET BASIC ROTATION Setting any basic rotation.	486

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 α (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**.



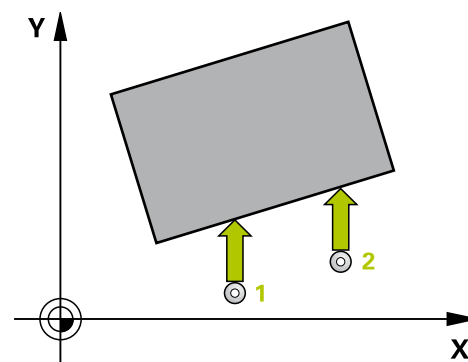
14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

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 467) 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:

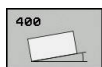


Before a cycle definition you must have programmed a tool call to define the touch probe axis.

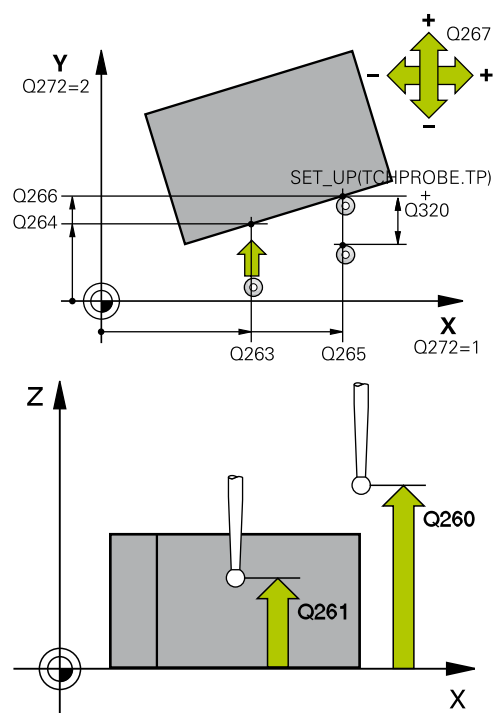
The TNC will reset an active basic rotation at the beginning of the cycle.

BASIC ROTATION (Cycle 400, DIN/ISO: G400) 14.2

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
 - 1: Reference axis = measuring axis
 - 2: Minor axis = measuring axis
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
 - 1: Negative Traverse direction
 - +1: Positive traverse direction
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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 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 PNT IN 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 ROTATION ANG.
Q305=0	;NUMBER IN TABLE

14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

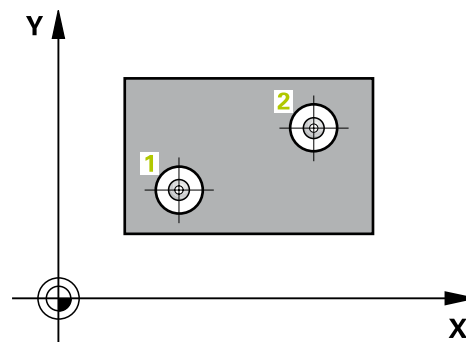
- ▶ **Q307 Preset value for rotation angle** (absolute):
If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- ▶ **Q305 Preset number in table?**: 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

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 467) 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.

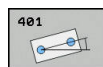
If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

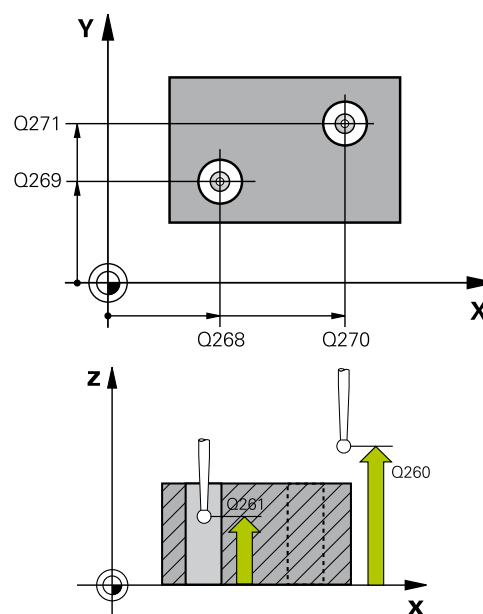
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.3 BASIC ROTATION over two holes (Cycle 401, DIN/ISO: G401)

Cycle parameters



- ▶ **Q268 1st hole: center in 1st axis?** (absolute):
Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q269 1st hole: center in 2nd axis?** (absolute):
Center of the first hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q270 2nd hole: center in 1st axis?** (absolute):
Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q271 2nd hole: center in 2nd axis?** (absolute):
Center of the second hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q307 Preset value for rotation angle** (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
- ▶ **Q305 Preset number in table?:** 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 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 ROTATION ANG.
Q305=0	;NUMBER IN TABLE
Q402=0	;COMPENSATION
Q337=0	;SET TO ZERO

BASIC ROTATION over two holes (Cycle 401, DIN/ISO: G401) 14.3

- ▶ **Q402 Basic rotation/alignment (0/1):** 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 tableIf 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**.
- ▶ **Q337 Set to zero after alignment?:** 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**.

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

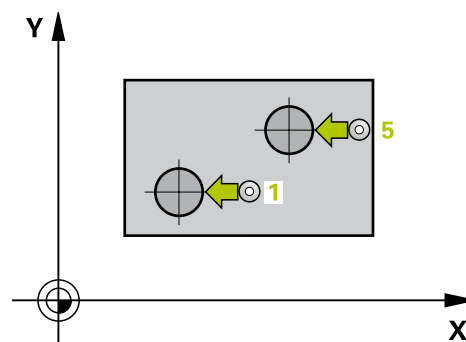
14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402)

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 467) 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:

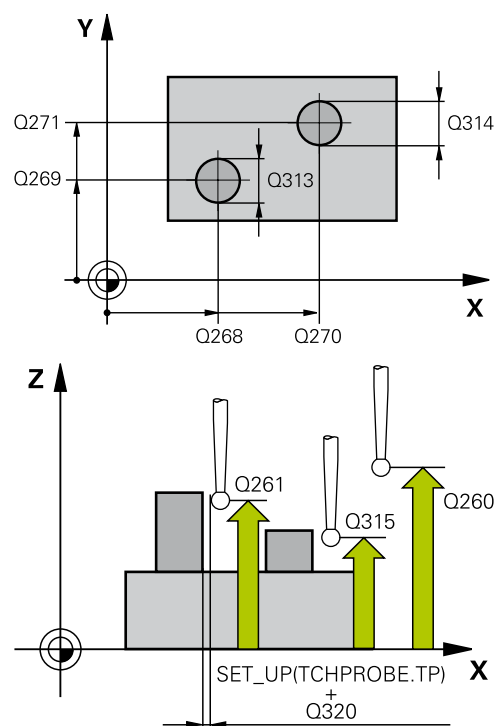
- C for tool axis Z
- B for tool axis Y
- A for tool axis X

BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402) 14.4

Cycle parameters



- ▶ **Q268 1st stud: center in 1st axis?** (absolute): Center of the first stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q269 1st stud: center in 2nd axis?** (absolute): Center of the first stud in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q313 Diameter of stud 1?** Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- ▶ **Q261 Meas. height stud 1 in TS axis?** (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
- ▶ **Q270 2nd stud: center in 1st axis?** (absolute): Center of the second stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q271 2nd stud: center in 2nd axis?** (absolute): Center of the second stud in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q313 Diameter of stud 2?** Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- ▶ **Q315 Meas. height stud 2 in TS axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q313=60	;DIAMETER OF STUD 1
Q261=-5	;MEAS. HEIGHT STUD 1
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q314=60	;DIAMETER OF STUD 2
Q315=-5	;MEAS. HEIGHT STUD 2
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q307=0	;PRESET ROTATION ANG.
Q305=0	;NUMBER IN TABLE
Q402=0	;COMPENSATION
Q337=0	;SET TO ZERO

14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402)

- ▶ **Q307 Preset value for rotation angle** (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

- ▶ **Q305 Preset number in table?**: 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

- ▶ **Q402 Basic rotation/alignment (0/1)**: 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**.

- ▶ **Q337 Set to zero after alignment?**: 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**.

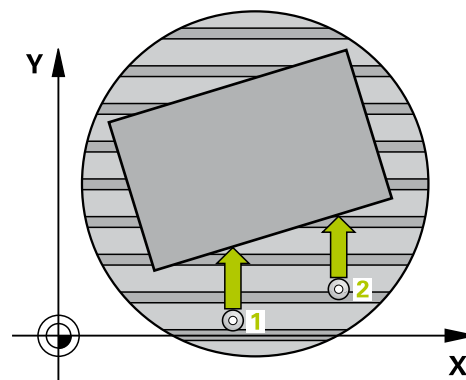
BASIC ROTATION compensation via rotary axis (Cycle 403, 14.5 DIN/ISO: G403)

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 467) 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 $+90^\circ$. 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**.

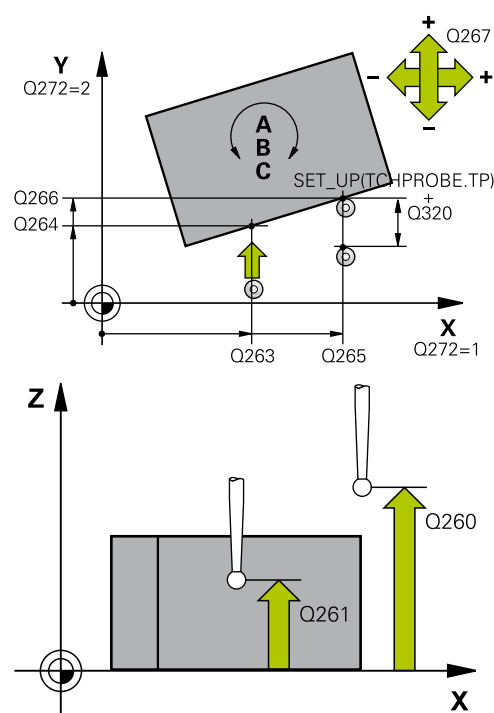
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403)

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: 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
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
-1: Negative Traverse direction
+1: Positive traverse direction
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool 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 PNT IN 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

BASIC ROTATION compensation via rotary axis (Cycle 403, 14.5 DIN/ISO: G403)

- ▶ **Q301 Move to clearance height (0/1)?:** 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
- ▶ **Q312 Axis for compensating movement?:**
 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 axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting.
4: Compensate misalignment with rotary axis A
5: Compensate misalignment with rotary axis B
6: Compensate misalignment with rotary axis C
- ▶ **Q337 Set to zero after alignment?:** 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
- ▶ **Q305 Number in table?** 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
- ▶ **Q303 Meas. value transfer (0,1)?:** 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).
- ▶ **Q380 Ref. angle in ref. axis?:** Angle with which the TNC is to align the probed straight line. Only effective if the rotary axis = Automatic mode or C is selected (Q312 = 0 or 6). Input range -360.000 to 360.000

Q301=0	;MOVE TO CLEARANCE
--------	--------------------

Q312=0	;COMPENSATION AXIS
--------	--------------------

Q337=0	;SET TO ZERO
--------	--------------

Q305=1	;NUMBER IN TABLE
--------	------------------

Q303=+1	;MEAS. VALUE TRANSFER
---------	-----------------------

Q380=+90	;REFERENCE ANGLE
----------	------------------

14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)

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.

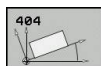
NC blocks

5 TCH PROBE 404 SET BASIC ROTATION

Q307=+0 ;PRESET ROTATION
ANG.

Q305=-1 ;NUMBER IN TABLE

Cycle parameters



- ▶ **Q307 Preset value for rotation angle:** Angular value at which the basic rotation is to be set. Input range -360.000 to 360.000
- ▶ **Q305 Preset number in table?:** 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 places 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

Compensating workpiece misalignment by rotating the C axis 14.7 (Cycle 405, DIN/ISO: G405)

14.7 Compensating workpiece misalignment by rotating the C axis (Cycle 405, DIN/ISO: G405)

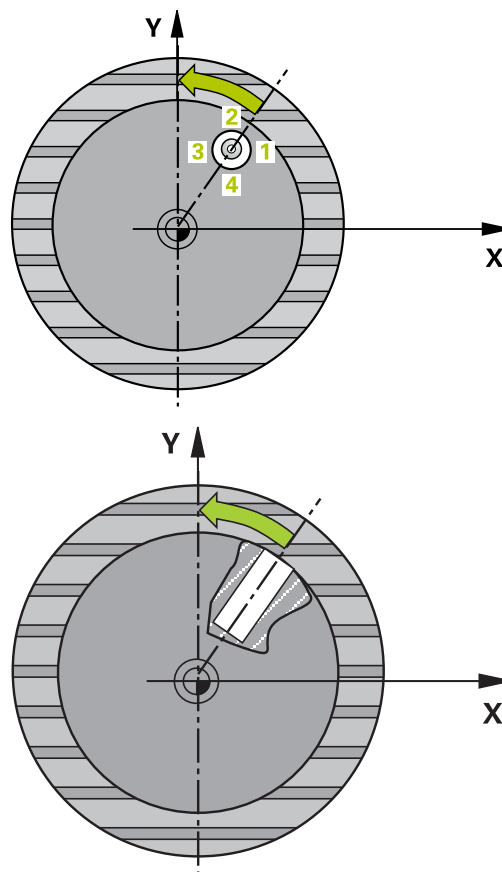
Cycle run

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 467) 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 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.



Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.7 Compensating workpiece misalignment by rotating the C axis (Cycle 405, DIN/ISO: G405)

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.

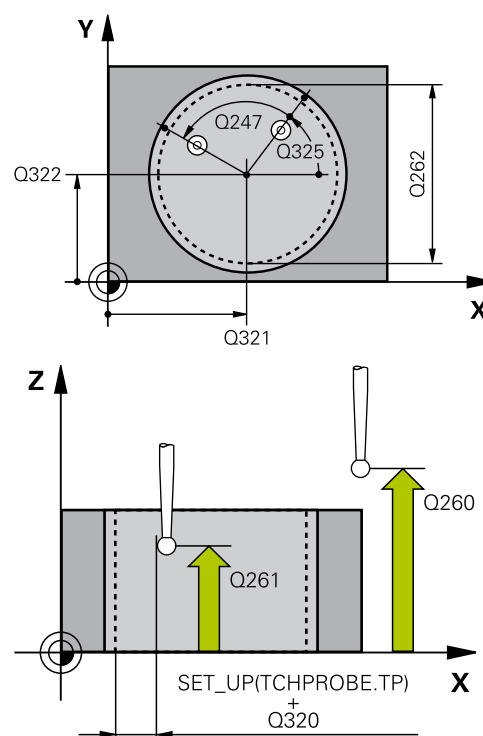
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the circle center. Minimum input value: 5°.

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the hole in the secondary 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
- ▶ **Q262 Nominal diameter?**: 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
- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- ▶ **Q247 Intermediate 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
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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



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

Compensating workpiece misalignment by rotating the C axis 14.7 (Cycle 405, DIN/ISO: G405)

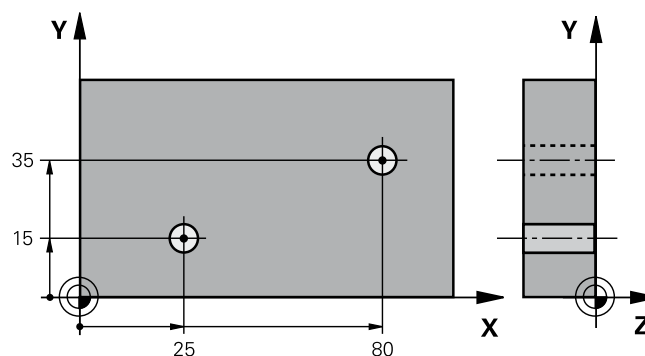
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
 - 0**: Move at measuring height between measuring points
 - 1**: Move at clearance height between measuring points
- ▶ **Q337 Set to zero after alignment?**: 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.

Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q337=0	;SET TO ZERO

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.8 Example: Determining a basic rotation from two holes

14.8 Example: Determining a basic rotation from two holes



0 BEGIN P GM 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 ROTATION ANG.	Angle of the reference line
Q305=0	;NUMBER IN TABLE	
Q402=1	;COMPENSATION	Compensate misalignment by rotating the rotary table
Q337=1	;SET TO ZERO	Set the display to zero after the alignment
3 CALL PGM 35K47		
4 END PGM CYC401 MM		

15

**Touch Probe
Cycles: Automatic
Datum Setting**

15.1 Fundamentals

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.

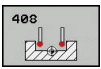




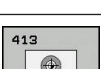

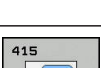
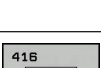

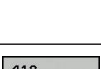



The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

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

Soft key	Cycle	Page
	408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as datum	496
	409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as datum	500
	410 DATUM INSIDE RECTANGLE Measuring the inside length and width of a rectangle, and defining the center as datum	503
	411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as datum	507
	412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum	510
	413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum	515
	414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as datum	519
	415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as datum	524
	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	528
	417 DATUM IN TS AXIS (2nd soft-key level) Measuring any position in the touch probe axis and defining it as datum	532
	418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum	534
	419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as datum	538

15.1 Fundamentals

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	Set reference point 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.

Touch Probe Cycles: Automatic Datum Setting

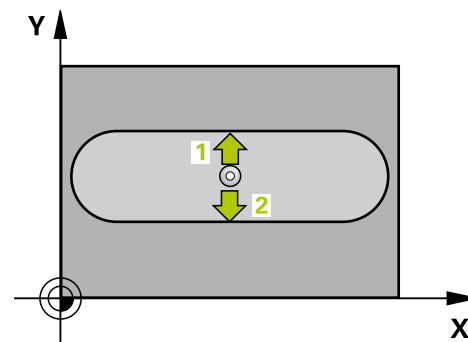
15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408)

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 467) 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 "Characteristics common to all touch probe cycles for datum setting", page 494) 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.

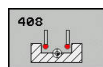
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

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.

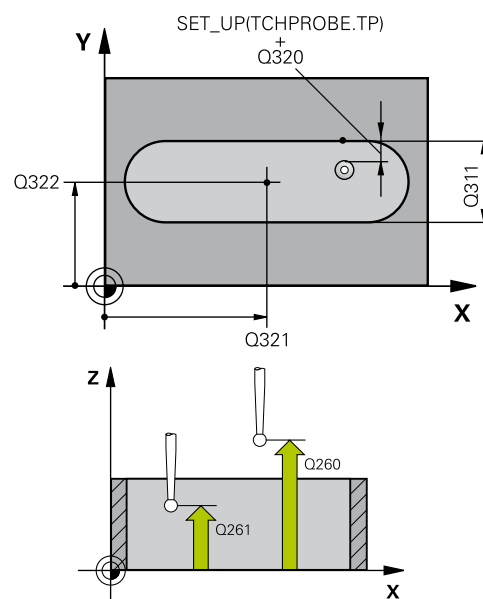
Touch Probe Cycles: Automatic Datum Setting

15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408)

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the slot in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the slot in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q311 Width of slot?** (incremental): Width of the slot, regardless of its position in the working plane. Input range 0 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
 - 1: Reference axis = measuring axis
 - 2: Minor axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q305 Number in table?**: 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
- ▶ **Q405 New datum?** (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
- ▶ **Q303 Meas. value transfer (0,1)?**: 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	;NUMBER 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

DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408) 15.2

- ▶ **Q381 Probe in TS axis? (0/1):** 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis? (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis? (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis? (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
- ▶ **Q333 New datum in TS axis? (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

Touch Probe Cycles: Automatic Datum Setting

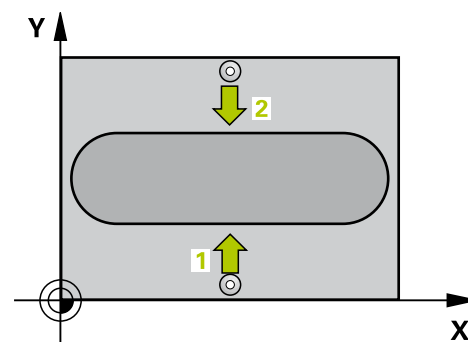
15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409)

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 467) 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 494) 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.

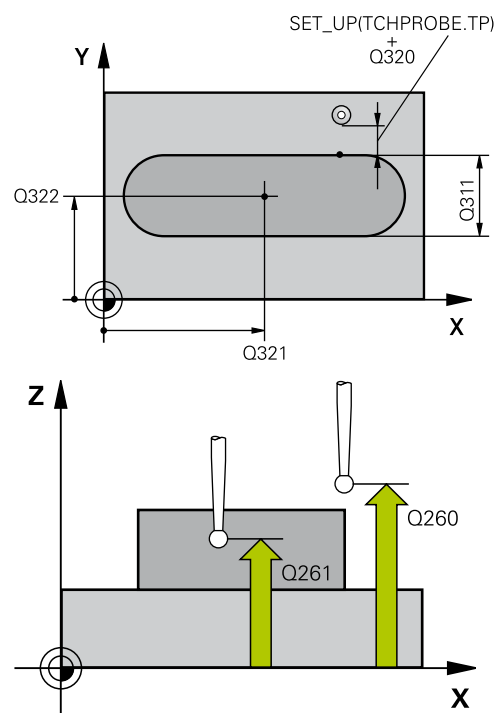
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409) 15.3

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the ridge in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the ridge in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q311 Ridge width?** (incremental): Width of the ridge, regardless of its position in the working plane. Input range 0 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
 - 1: Reference axis = measuring axis
 - 2: Minor axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?**: 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
- ▶ **Q405 New datum?** (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
- ▶ **Q303 Meas. value transfer (0,1)?**: 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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 RIDGE CENTER REF PT	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q311=25	;RIDGE WIDTH
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q305=10	;NUMBER 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

Touch Probe Cycles: Automatic Datum Setting

15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409)

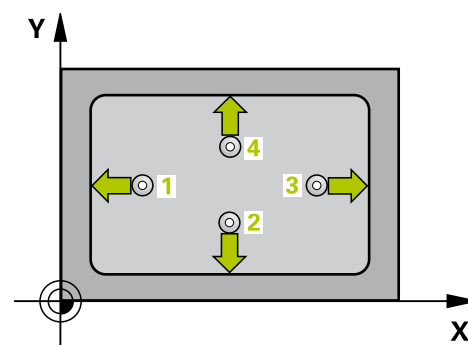
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):
Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (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 467) 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 494).
- 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

Touch Probe Cycles: Automatic Datum Setting

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

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.

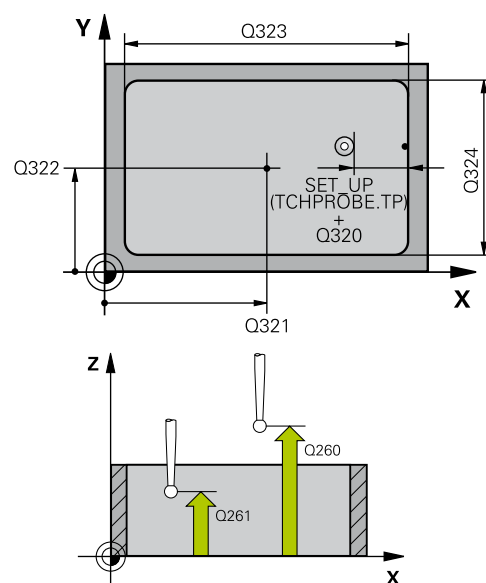
If you set a datum ($Q303 = 0$) with the touch probe cycle and also use probe in TS axis ($Q381 = 1$), then no coordinate transformation must be active.

DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410) 15.4

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q323 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane Input range 0 to 99999.9999
- ▶ **Q324 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane Input range 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q305 Number in table?**: 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 on the pocket center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- ▶ **Q331 New datum in reference axis?** (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
- ▶ **Q332 New datum in minor axis?** (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 ;NUMBER 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

Touch Probe Cycles: Automatic Datum Setting

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
0: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (absolute): Coordinate at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

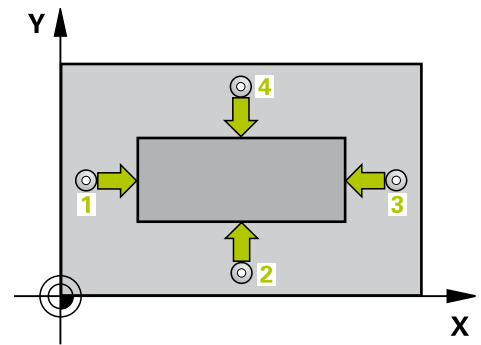
DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411) 15.5

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.

- Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from **FMAX** column) (see "Executing touch probe cycles", page 467) 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.
- Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column **F**).
- 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.
- The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
- 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 494).
- 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.

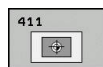
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

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.

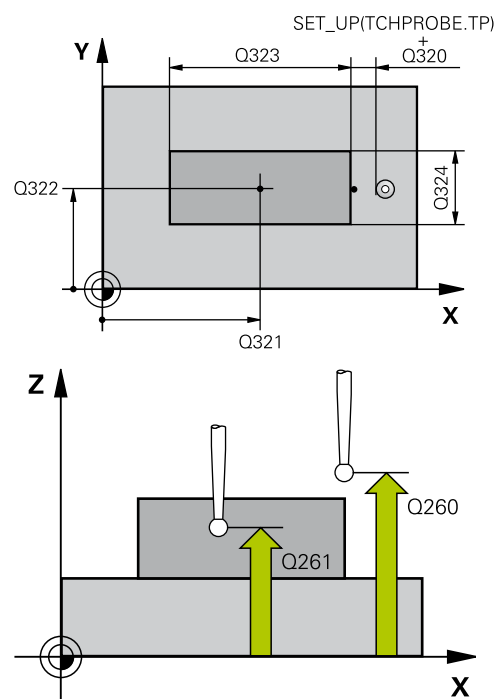
Touch Probe Cycles: Automatic Datum Setting

15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411)

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q323 First side length?** (incremental): Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q324 Second side length?** (incremental): Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q305 Number in table?**: 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
- ▶ **Q331 New datum in reference axis?** (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
- ▶ **Q332 New datum in minor axis?** (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



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	;NUMBER 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

DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411) 15.5

- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
0: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (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

Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;DATUM

Touch Probe Cycles: Automatic Datum Setting

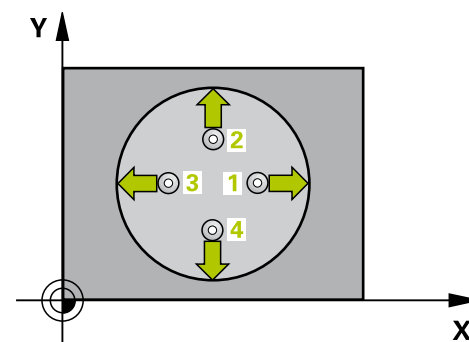
15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

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 467) 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 494) 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

DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412) 15.6

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°.

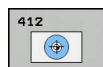
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

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.

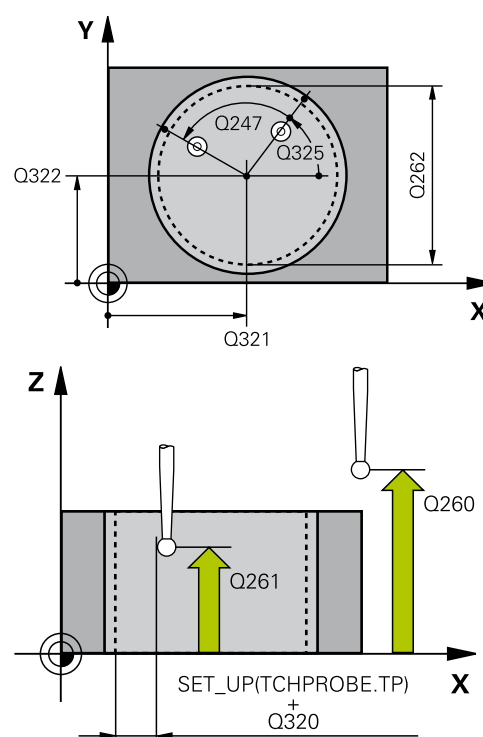
Touch Probe Cycles: Automatic Datum Setting

15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary 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
- ▶ **Q262 Nominal diameter?**: 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
- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- ▶ **Q247 Intermediate 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
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q305 Number in table?**: 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 on the pocket 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 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	;NUMBER 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

DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412) 15.6

- ▶ **Q331 New datum in reference axis?** (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
- ▶ **Q332 New datum in minor axis?** (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
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
 - 0**: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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

Touch Probe Cycles: Automatic Datum Setting

15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

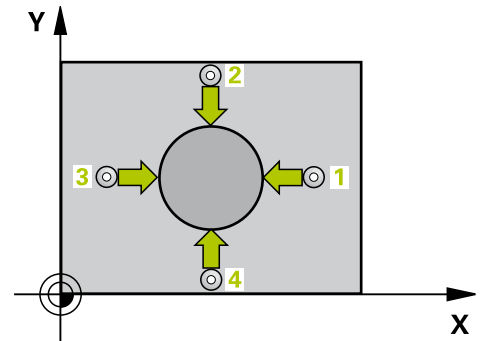
- ▶ **Q333 New datum in TS axis?** (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the TNC should measure the stud with 4 or 3 probing points:
 - 4**: Use 4 measuring points (default setting)
 - 3**: Use 3 measuring points
- ▶ **Q365 Type of traverse? Line=0/arc=1**: 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 467) 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 494) 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.

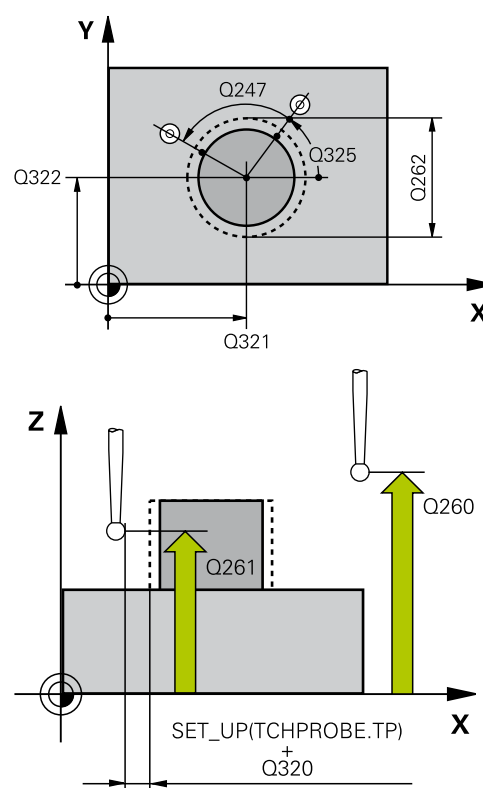
Touch Probe Cycles: Automatic Datum Setting

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)

Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary 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
- ▶ **Q262 Nominal diameter?**: 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
- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- ▶ **Q247 Intermediate 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
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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 ;NUMBER IN TABLE

Q331=+0 ;DATUM

Q332=+0 ;DATUM

DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413) 15.7

- ▶ **Q305 Number in table?:** 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
- ▶ **Q331 New datum in reference axis?** (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
- ▶ **Q332 New datum in minor axis?** (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
- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined reference point 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 494)
 - 0:** Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1):** 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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

Touch Probe Cycles: Automatic Datum Setting

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)

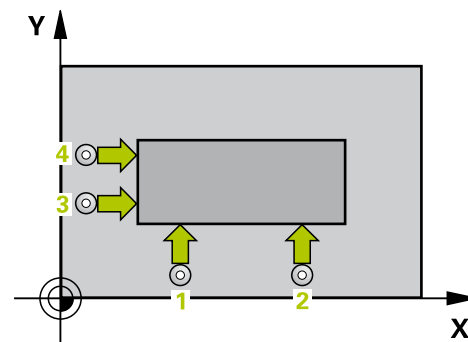
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q333 New datum in TS axis?** (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q423 No. probe points in plane (4/3)?**: 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
- ▶ **Q365 Type of traverse? Line=0/arc=1**: 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 467) 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 494) 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

Touch Probe Cycles: Automatic Datum Setting

15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414)

Please note while programming:

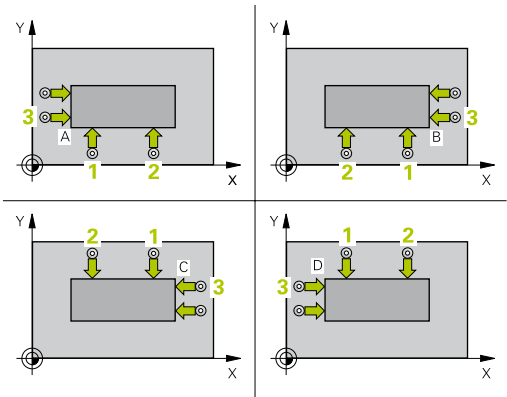


Danger of collision!

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).



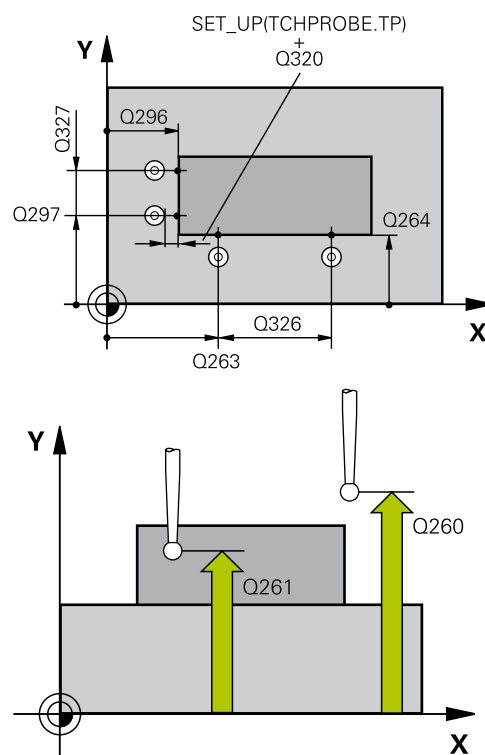
Corner	X coordinate	Y coordinate
A	Point 1 greater than point 3	Point 1 less than point 3
B	Point 1 less than point 3	Point 1 less than point 3
C	Point 1 less than point 3	Point 1 greater than point 3
D	Point 1 greater than point 3	Point 1 greater than point 3

DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414) 15.8

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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 PNT IN 1ST AXIS
Q297=+25	;3RD PNT IN 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	;NUMBER IN TABLE

Touch Probe Cycles: Automatic Datum Setting

15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414)

- ▶ **Q304 Execute basic rotation (0/1)?**: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
0: Do not execute basic rotation
1: Execute basic rotation
- ▶ **Q305 Number in table?**: Enter the number in the datum/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
- ▶ **Q331 New datum in reference axis?** (absolute):
 Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q332 New datum in minor axis?** (absolute):
 Coordinate in the minor axis at which the TNC should set the corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
0: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999

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

DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414) 15.8

- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (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

Touch Probe Cycles: Automatic Datum Setting

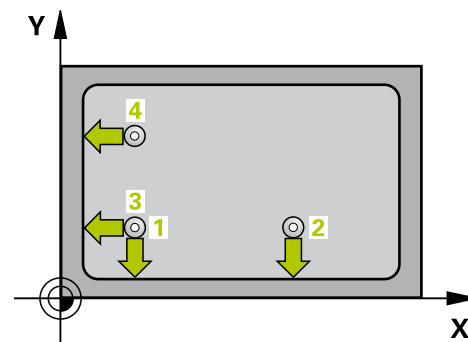
15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415)

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 467) 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 494) 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

DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415) 15.9

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.

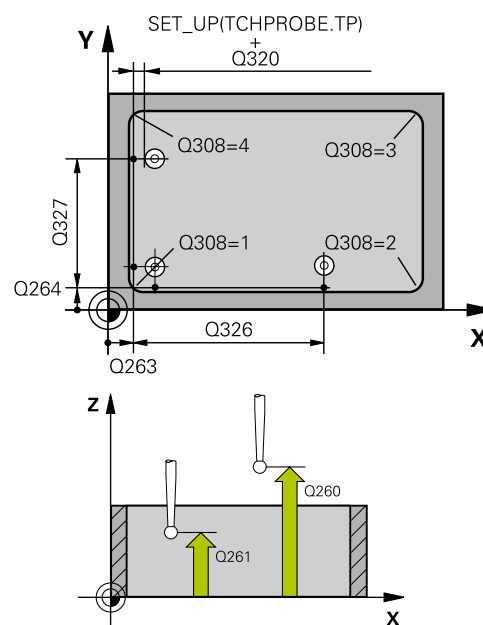
Touch Probe Cycles: Automatic Datum Setting

15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415)

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q308 Corner? (1/2/3/4):** Number identifying the corner which the TNC is to set as datum. Input range 1 to 4
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?:** 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
- ▶ **Q304 Execute basic rotation (0/1)?:** Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
0: Do not execute basic rotation
1: Execute basic rotation
- ▶ **Q305 Number in table?:** Enter the number in the datum/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

Q327=45 ;SPACING IN 2ND AXIS

Q308=+1 ;CORNER

Q261=-5 ;MEASURING HEIGHT

Q320=0 ;SET-UP CLEARANCE

Q260=+20 ;CLEARANCE HEIGHT

Q301=0 ;MOVE TO CLEARANCE

Q304=0 ;BASIC ROTATION

Q305=7 ;NUMBER 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

DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415) 15.9

- ▶ **Q331 New datum in reference axis?** (absolute):
Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q332 New datum in minor axis?** (absolute):
Coordinate in the minor axis at which the TNC should set the corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
 0: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (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

Touch Probe Cycles: Automatic Datum Setting

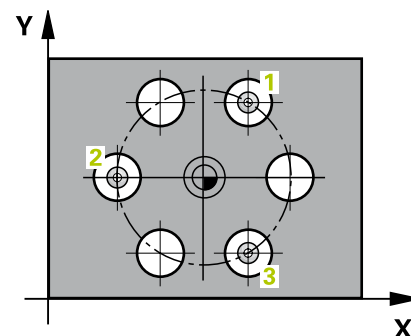
15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)

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 467) 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 494) 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

DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416) 15.10

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.

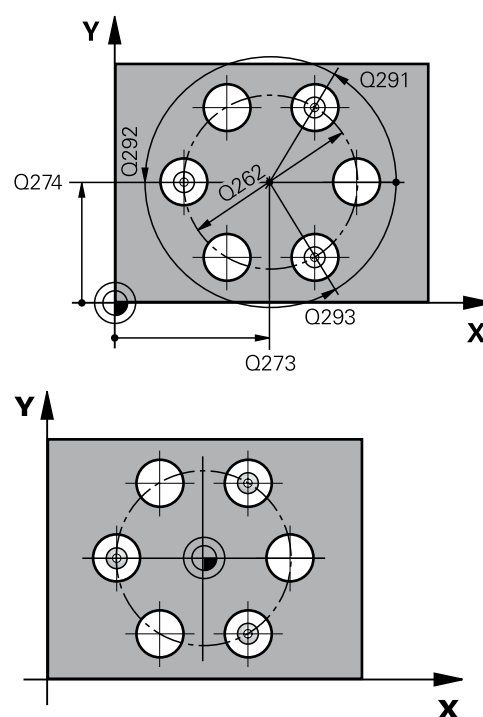
Touch Probe Cycles: Automatic Datum Setting

15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)

Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: 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
- ▶ **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- ▶ **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- ▶ **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?**: Enter the number in the datum/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 in 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
- ▶ **Q331 New datum in reference axis?** (absolute): Coordinate in the reference axis at which the TNC should set the bolt-hole circle center. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q332 New datum in minor axis?** (absolute): Coordinate in the minor axis at which the TNC should set the bolt-hole circle 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 ;NUMBER 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

DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416) 15.10

- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
 0: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (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
- ▶ **Q320 Set-up clearance?** (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

Touch Probe Cycles: Automatic Datum Setting

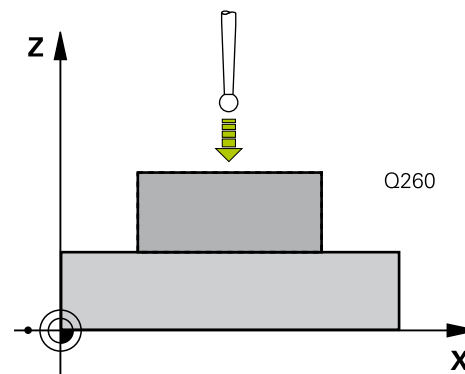
15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417)

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 467) 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 494) 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!

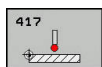
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.



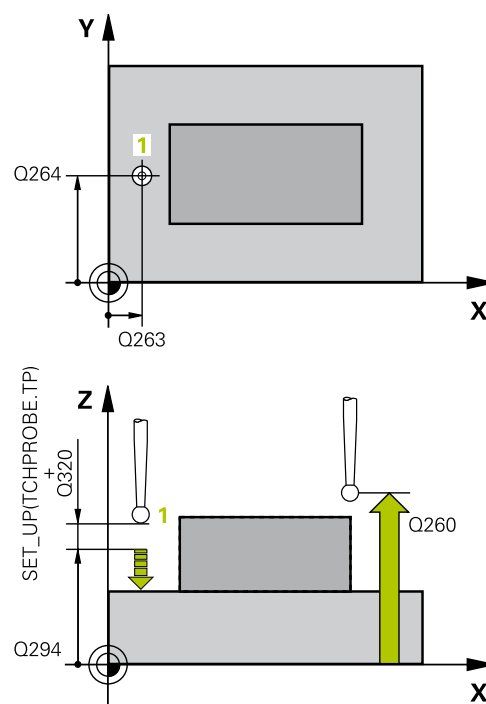
Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC then sets the datum in this axis.

DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417) 15.11

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?**: Enter the number in the datum/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
- ▶ **Q333 New datum in TS axis?** (absolute): Coordinate at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
 - 0: Write the measured datum into 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	;NUMBER IN TABLE
Q333=+0	;DATUM
Q303=+1	;MEAS. VALUE TRANSFER

Touch Probe Cycles: Automatic Datum Setting

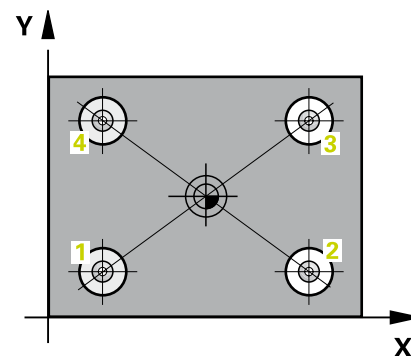
15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)

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 467) 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 494). 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

DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418) 15.12

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.

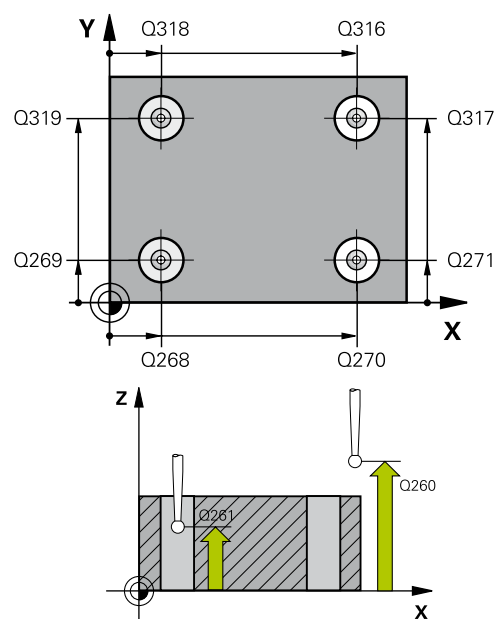
Touch Probe Cycles: Automatic Datum Setting

15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)

Cycle parameters



- ▶ **Q268 1st hole: center in 1st axis?** (absolute):
Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q269 1st hole: center in 2nd axis?** (absolute):
Center of the first hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q270 2nd hole: center in 1st axis?** (absolute):
Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q271 2nd hole: center in 2nd axis?** (absolute):
Center of the second hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q316 3rd hole: Center in 1st axis?** (absolute):
Center of the third hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q317 3rd hole: Center in 2nd axis?** (absolute):
Center of the third hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q318 4th hole: Center in 1st axis?** (absolute):
Center of the fourth hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q319 4th hole: Center in 2nd axis?** (absolute):
Center of the fourth hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?:** Enter the number in the datum/preset table in which the TNC is to save the coordinates of the intersection of the connecting lines. 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
- ▶ **Q331 New datum in reference axis?** (absolute):
Coordinate in the reference axis at which the TNC should set the 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 ;NUMBER 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	

DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418) 15.12

- ▶ **Q332 New datum in minor axis?** (absolute):
Coordinate in the minor axis at which the TNC should set the intersection of the connecting lines. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
 0: Write the measured datum into 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).
- ▶ **Q381 Probe in TS axis? (0/1)**: 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
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (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 = 1. Input range -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (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
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (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
- ▶ **Q333 New datum in TS axis?** (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

Touch Probe Cycles: Automatic Datum Setting

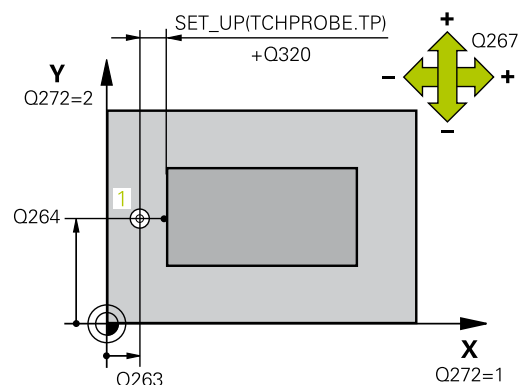
15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

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 467) 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 494).



Please note while programming:



Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you want to save the datum of several axes in the preset table, you can use Cycle 419 several times in a row. However, you also have to reactivate the preset number after every run of Cycle 419. If you work with preset 0 as active preset, this process is not required.

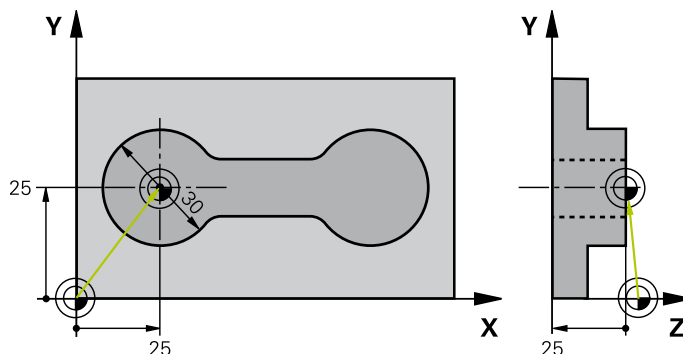
Touch Probe Cycles: Automatic Datum Setting

15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

- ▶ **Q333 New datum in TS axis?** (absolute): Coordinate at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined reference point 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 494)
 - 0**: Write the measured datum into 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).

Example: Datum setting in center of a circular segment and on top 15.14 surface of workpiece

15.14 Example: Datum setting in center of a circular segment and on top surface of workpiece



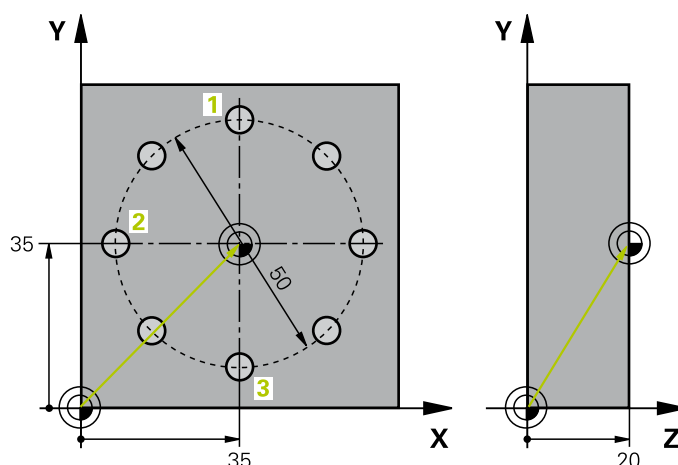
0 BEGIN PGM CYC413 MM		
1 TOOL CALL 69 Z		Call tool 0 to define the touch probe axis
2 TCH PROBE 413 DATUM OUTSIDE CIRCLE		
Q321=+25	;CENTER IN 1ST AXIS	Center of circle: X coordinate
Q322=+25	;CENTER IN 2ND AXIS	Center of circle: Y coordinate
Q262=30	;NOMINAL DIAMETER	Circle diameter
Q325=+90	;STARTING 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	;NUMBER 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		
4 END PGM CYC413 MM		

Touch Probe Cycles: Automatic Datum Setting

15.15 Example: Datum setting on top surface of workpiece and in center of a bolt hole circle

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 POBE 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 ;NUMBER 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 ;NUMBER IN TABLE		Enter center of bolt hole circle (X and Y) in line 1
Q331=+0 ;DATUM		
Q332=+0 ;DATUM		

Example: Datum setting on top surface of workpiece and in center 15.15 of a bolt hole circle

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

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.

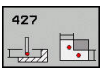
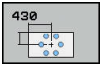
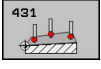


The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

Refer to your machine manual.

The TNC offers twelve cycles for measuring workpieces automatically.

Soft key	Cycle	Page
	0 REFERENCE PLANE Measuring a coordinate in a selectable axis	552
	1 POLAR DATUM PLANE Measuring a point in a probing direction	553
	420 MEASURE ANGLE Measuring an angle in the working plane	554
	421 MEASURE HOLE Measuring the position and diameter of a hole	557
	422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud	562
	423 MEASURE RECTANGLE INSIDE Measuring the position, length and width of a rectangular pocket	567
	424 MEASURE RECTANGLE OUTSIDE Measuring the position, length and width of a rectangular stud	570
	425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width	573
	426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge	576

Soft key	Cycle	Page
	427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis	579
	430 MEASURE BOLT HOLE CIRCLE (2nd soft-key row) Measuring position and diameter of a bolt hole circle	582
	431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane	585

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. The TNC will save the file in the directory that also contains the associated NC program.



Use the HEIDENHAIN data transfer software TNCRemo if you wish to output the measuring log over the data interface.

16.1 Fundamentals

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

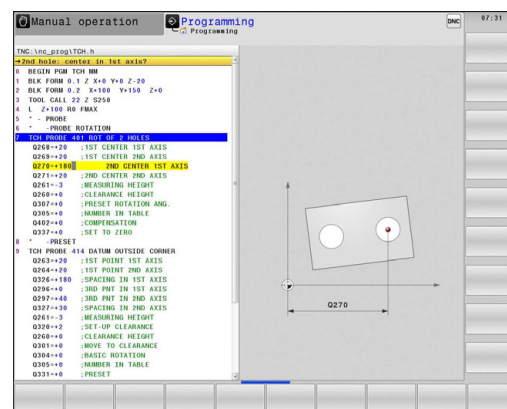
Further measuring results: Measuring height: -5.0000

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.

16.1 Fundamentals

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.

Milling tool: If you reference parameter Q330 to a milling tool, the appropriate values are compensated in the following way: the TNC basically always compensates the tool radius in column DR of the tool table, even if the measured deviation is within the stated tolerance. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: must be reworked).

Turning tool: (only applies to Cycles 421, 422, 427) If you reference parameter Q330 to a turning tool, the appropriate values in row DZL and DXL, respectively, are compensated. The TNC monitors the breakage tolerance which is defined in column LBREAK. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: must be reworked).

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.

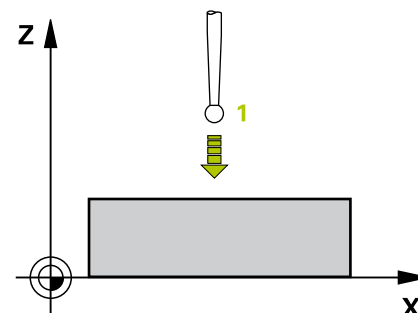
Touch Probe Cycles: Automatic Workpiece Inspection

16.2 DATUM PLANE (Cycle 0, DIN/ISO: G55)

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
- ▶ **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-
```

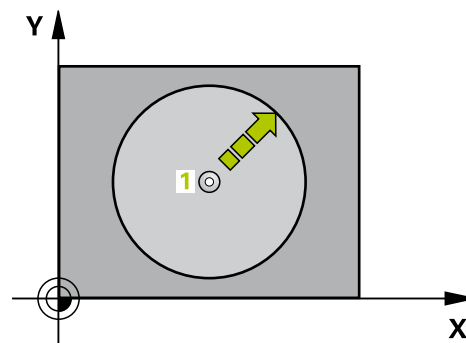
```
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

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!

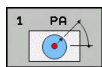
Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.



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
- ▶ **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 DATUM

68 TCH PROBE 1.1 X ANGLE: +30

69 TCH PROBE 1.2 X+5 Y+0 Z-5

Touch Probe Cycles: Automatic Workpiece Inspection

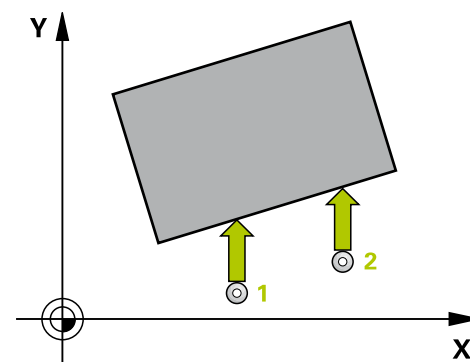
16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

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 The TNC positions the touch probe at rapid traverse (value from **FMAX** column) following the positioning logic (see "Executing touch probe cycles", page 467) to the starting 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:



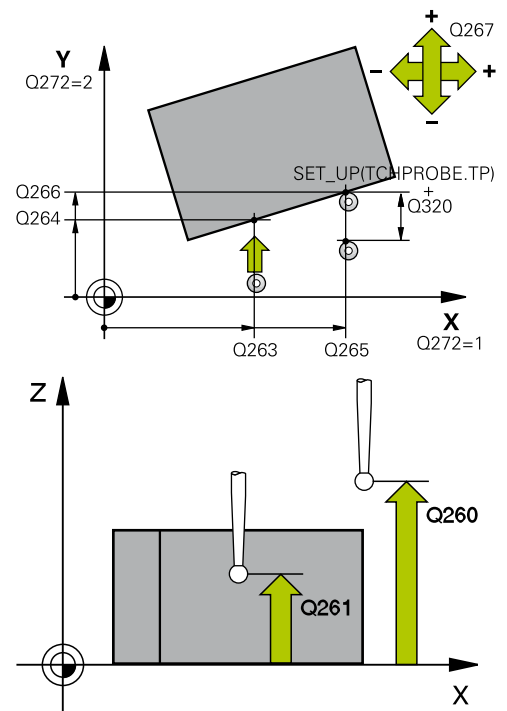
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

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



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: 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
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
 - 1**: Negative Traverse direction
 - +1**: Positive traverse direction
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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



NC blocks

5 TCH PROBE 420 MEASURE ANGLE	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+10	;1ST POINT 2ND AXIS
Q265=+15	;2ND PNT IN 1ST AXIS
Q266=+95	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT

16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q281 Measuring log (0/1/2)?**: 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.

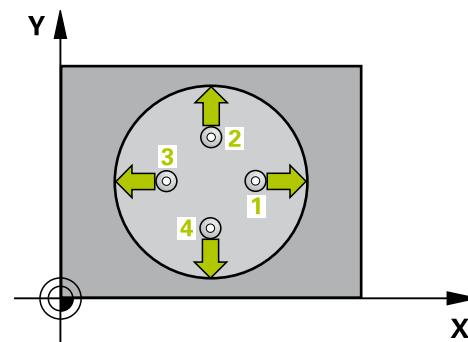
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q281=1	;MEASURING LOG

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 467) 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

16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)**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°

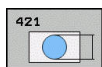
If you reference parameter Q330 to a turning tool, the following applies:

- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the TNC compensates the turning tool, the appropriate values in line DZL and DXL, respectively, are compensated.
- The TNC monitors the breakage tolerance which is defined in column LBREAK

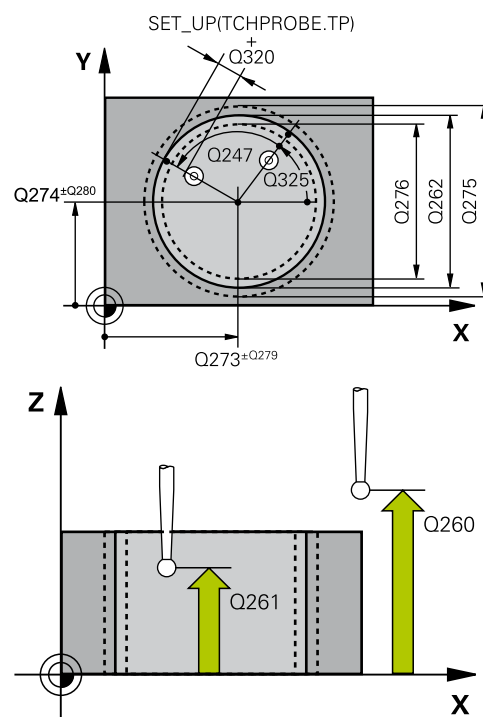
If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect

MEASURE HOLE (Cycle 421, DIN/ISO: G421) 16.5

Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the hole in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- ▶ **Q247 Intermediate 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
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 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

16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)

- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q275 Maximum limit of size for hole?**: Maximum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- ▶ **Q276 Minimum limit of size?**: Minimum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: Definition of whether TNC should create a measuring log:
0: Create no measuring log
1: Create measuring log: The TNC will save the **log file TCHPR421.TXT** by default in the directory that also contains the associated NC program.
2: Interrupt the program run and display the measuring log on the TNC screen. Resume program run with NC Start.
- ▶ **Q309 PGM stop if tolerance exceeded?**: 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

Q260=+20	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q275=75.12	MAXIMUM LIMIT
Q276=74.95	MINIMUM LIMIT
Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

- ▶ **Q330 Tool for monitoring?**: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
 - 0**: Monitoring inactive
 - >0**: Number or name of the tool that the TNC used for machining. You are able to apply a tool via soft key directly from the tool table.
- ▶ **Q423 No. probe points in plane (4/3)?**: 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
- ▶ **Q365 Type of traverse? Line=0/arc=1**: 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
- ▶ **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the TNC requires the exact working condition. Therefore enter the following:
 - 1**: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool** Q498=1
 - 0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification by e.g. Cycle 800 and parameter **Reverse the tool** Q498=0
- ▶ **Q531 Angle of incidence?**: Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence between the turning tool and the work piece during machining, e.g. from Cycle 800 parameter **Angle of incidence?** Q531. Input range: -180° to +180°

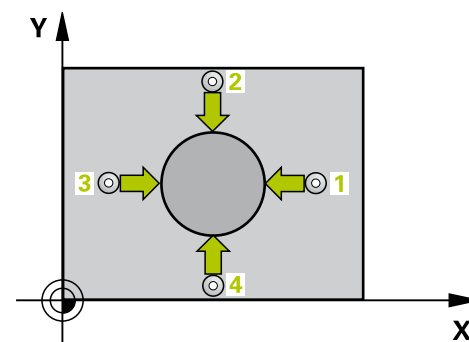
16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

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 467) 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°

If you reference parameter Q330 to a turning tool, the following applies:

- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the TNC compensates the turning tool, the appropriate values in line DZL and DXL, respectively, are compensated.
- The TNC monitors the breakage tolerance which is defined in column LBREAK

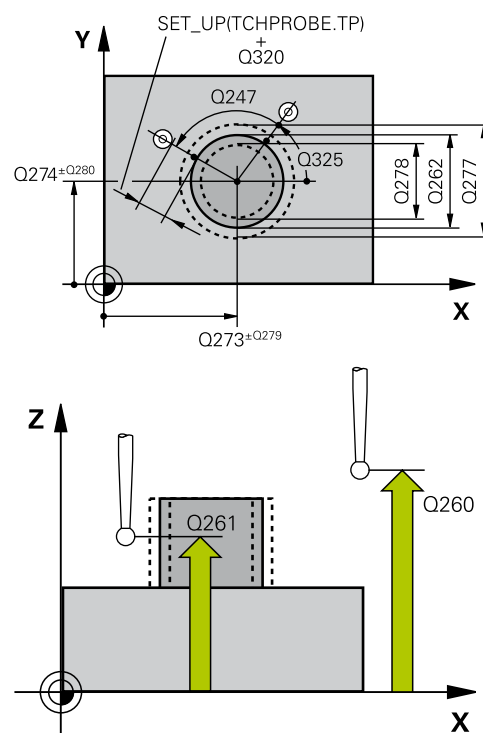
If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect

16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the diameter of the stud. Input range 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** (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
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q277 Maximum limit of size for stud?**: Maximum permissible diameter for the stud Input range 0 to 99999.9999
- ▶ **Q278 Minimum limit of size for stud?**: 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

Q277=35.15 ;MAXIMUM LIMIT

Q278=34.9 ;MINIMUM LIMIT

Q279=0.05 ;TOLERANCE 1ST CENTER

Q280=0.05 ;TOLERANCE 2ND CENTER

MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422) 16.6

- ▶ **Q279 Tolerance for center 1st axis?:** Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?:** Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?:** 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.
- ▶ **Q309 PGM stop if tolerance exceeded?:** 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
- ▶ **Q330 Tool for monitoring?:** Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). 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
- ▶ **Q423 No. probe points in plane (4/3)?:** 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

Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

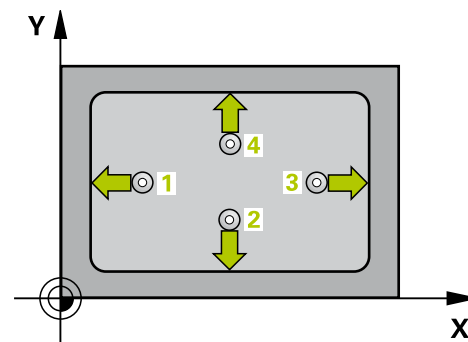
- ▶ **Q365 Type of traverse? Line=0/arc=1:** Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
 - 0:** Move in a straight line between machining operations
 - 1:** Move in a circular arc on the pitch circle diameter between machining operations
- ▶ **Q498 Reverse tool (0=no/1=yes)?:** Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the TNC requires the exact working condition. Therefore enter the following:
 - 1:** Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool** Q498=1
 - 0:** Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification by e.g. Cycle 800 and parameter **Reverse the tool** Q498=0
- ▶ **Q531 Angle of incidence?:** Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence between the turning tool and the work piece during machining, e.g. from Cycle 800 parameter **Angle of incidence?** Q531. Input range: -180° to +180°

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 467) 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.

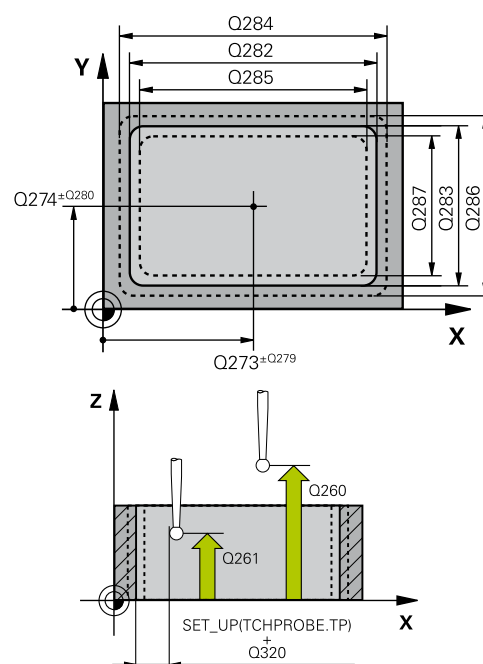
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

16.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423)

Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the pocket in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q282 1st side length (nominal value)?**: Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q283 2nd side length (nominal value)?**: Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q284 Max. size limit 1st side length?**: Maximum permissible length of the pocket. Input range 0 to 99999.9999
- ▶ **Q285 Min. size limit 1st side length?**: Minimum permissible length of the pocket. Input range 0 to 99999.9999
- ▶ **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the pocket. Input range 0 to 99999.9999
- ▶ **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the pocket. Input range 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?**: 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 TOLERANCE
Q330=0	;TOOL

MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423) 16.7

- ▶ **Q281 Measuring log (0/1/2)?**: 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.
- ▶ **Q309 PGM stop if tolerance exceeded?**: 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
- ▶ **Q330 Tool for monitoring?**: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). 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

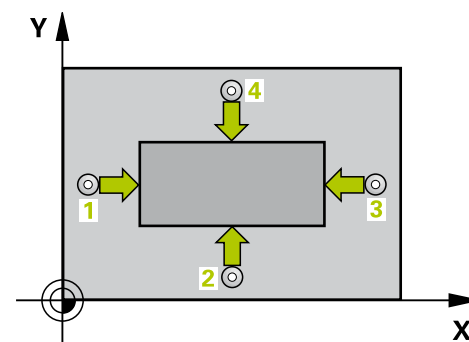
16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)

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 values in system parameters.

- 1 The TNC positions the touch probe at rapid traverse (value from **FMAX** column) following the positioning logic (see "Executing touch probe cycles", page 467) to the starting 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

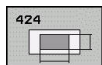
Please note while programming:



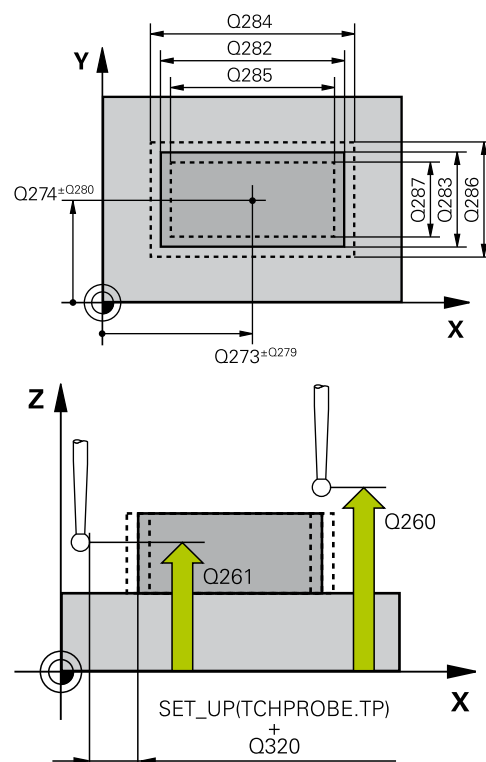
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424) 16.8

Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q282 1st side length (nominal value)?**: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q283 2nd side length (nominal value)?**: Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: 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
- ▶ **Q284 Max. size limit 1st side length?**: Maximum permissible length of the stud. Input range 0 to 99999.9999
- ▶ **Q285 Min. size limit 1st side length?**: Minimum permissible length of the stud. Input range 0 to 99999.9999
- ▶ **Q286 Max. size limit 2nd side length?**: Maximum 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 ;2ND CENTER 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

16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)

- ▶ **Q287 Min. size limit 2nd side length?:** Minimum permissible width of the stud. Input range 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?:** Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?:** Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?:** 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.
- ▶ **Q309 PGM stop if tolerance exceeded?:** 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
- ▶ **Q330 Tool for monitoring?:** Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
 - 0:** Monitoring inactive
 - >0:** Number or name of the tool that the TNC used for machining. You are able to apply a tool via soft key directly from the tool table.

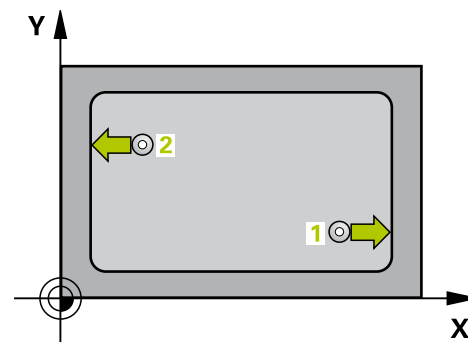
Q287=34.95	MIN. LIMIT 2ND SIDE
Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

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 467) 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

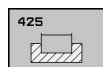
Please note while programming:



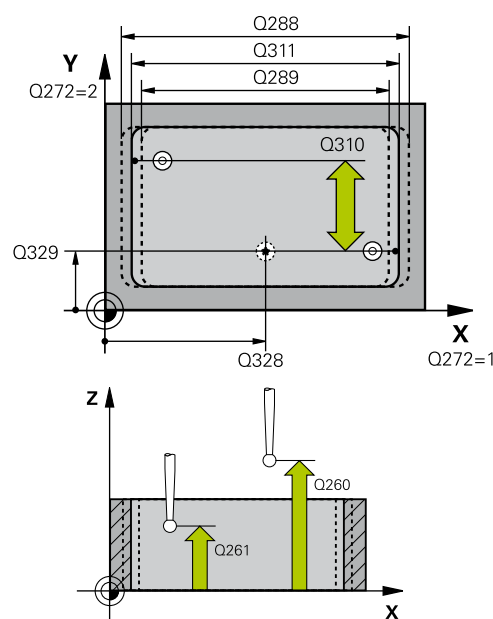
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)

Cycle parameters



- ▶ **Q328 Starting point in 1st axis?** (absolute): Starting point for probing in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q329 Starting point in 2nd axis?** (absolute): Starting point for probing in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q310 Offset for 2nd measurement (+/-)?** (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
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
 - 1: Reference axis = measuring axis
 - 2: Minor axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q311 Nominal length?**: Nominal value of the length to be measured. Input range 0 to 99999.9999
- ▶ **Q288 Maximum limit of size?**: Maximum permissible length. Input range 0 to 99999.9999
- ▶ **Q289 Minimum limit of size?**: 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.
- ▶ **Q309 PGM stop if tolerance exceeded?**: 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 LIMIT

Q289=25 ;MINIMUM LIMIT

Q281=1 ;MEASURING LOG

Q309=0 ;PGM STOP TOLERANCE

Q330=0 ;TOOL

Q320=0 ;SET-UP CLEARANCE

Q301=0 ;MOVE TO CLEARANCE

MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425) 16.9

- ▶ **Q330 Tool for monitoring?**: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
 - 0**: Monitoring inactive
 - >0**: Number or name of the tool that the TNC used for machining. You are able to apply a tool via soft key directly from the tool table.
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q301 Move to clearance height (0/1)?**: 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

Touch Probe Cycles: Automatic Workpiece Inspection

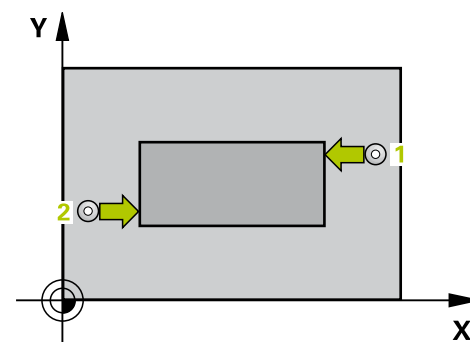
16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)

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 The TNC positions the touch probe at rapid traverse (value from **FMAX** column) following the positioning logic (see "Executing touch probe cycles", page 467) to the starting 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

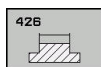
Please note while programming:



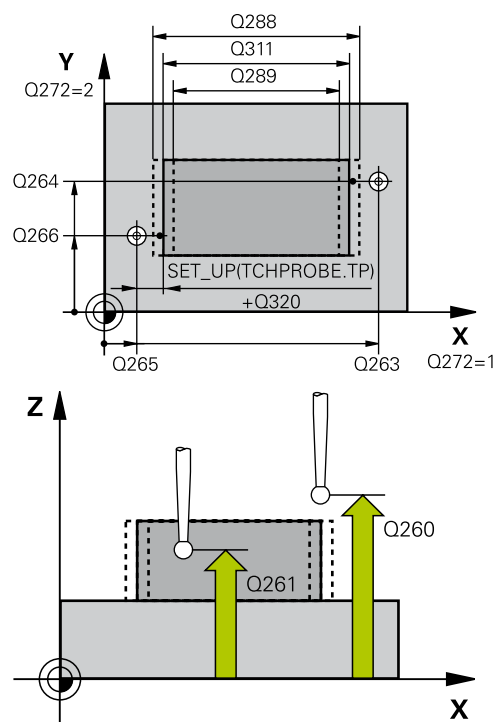
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426) 16.10

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
 - 1: Reference axis = measuring axis
 - 2: Minor axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q311 Nominal length?**: Nominal value of the length to be measured. Input range 0 to 99999.9999
- ▶ **Q288 Maximum limit of size?**: Maximum permissible length. Input range 0 to 99999.9999
- ▶ **Q289 Minimum limit of size?**: Minimum permissible length. Input range 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: 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 PNT IN 1ST AXIS

Q266=+85 ;2ND PNT IN 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 LIMIT

Q289=44.95 ;MINIMUM LIMIT

Q281=1 ;MEASURING LOG

Q309=0 ;PGM STOP TOLERANCE

Q330=0 ;TOOL

16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)

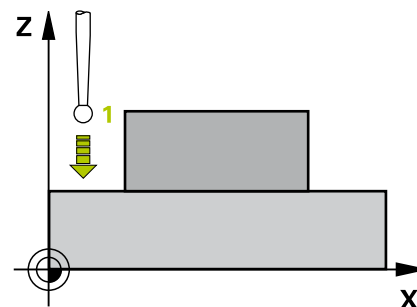
- ▶ **Q309 PGM stop if tolerance exceeded?:** Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
 - 0:** Do not interrupt program run, do not output an error message
 - 1:** Interrupt program run and output an error message
- ▶ **Q330 Tool for monitoring?:** Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
 - 0:** Monitoring inactive
 - >0:** Number or name of the tool that the TNC used for machining. You are able to apply a tool via soft key directly from the tool table.

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 467) 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.

If an axis of the active working plane is defined as measuring axis (Q272 = 1 or 2), the TNC compensates the tool radius. 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.

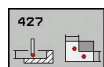
If you reference parameter Q330 to a turning tool, the following applies:

- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the TNC compensates the turning tool, the appropriate values in line DZL and DXL, respectively, are compensated.
- The TNC monitors the breakage tolerance which is defined in column LBREAK

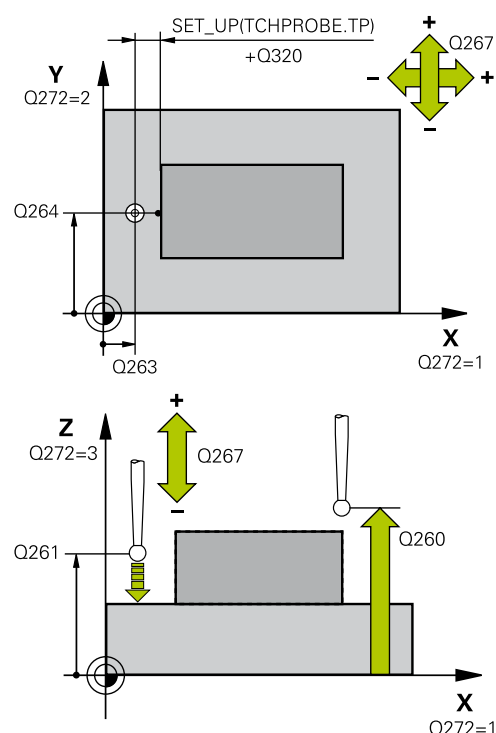
If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect

16.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: 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
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
 - 1: Negative Traverse direction
 - +1: Positive traverse direction
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: 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.



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 LIMIT

MEASURE COORDINATE (Cycle 427, DIN/ISO: G427) 16.11

- ▶ **Q288 Maximum limit of size?:** Maximum permissible measured value. Input range 0 to 99999.9999
- ▶ **Q289 Minimum limit of size?:** Minimum permissible measured value. Input range 0 to 99999.9999
- ▶ **Q309 PGM stop if tolerance exceeded?:** 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
- ▶ **Q330 Tool for monitoring?:** Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the TNC used for machining. You are able to apply a tool via soft key directly from the tool table.
- ▶ **Q498 Reverse tool (0=no/1=yes)?:** Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the TNC requires the exact working condition. Therefore enter the following:
1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool** Q498=1
0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification by e.g. Cycle 800 and parameter **Reverse the tool** Q498=0
- ▶ **Q531 Angle of incidence?:** Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence between the turning tool and the work piece during machining, e.g. from Cycle 800 parameter **Angle of incidence?** Q531. Input range: -180° to +180°

Q289=4.95 ;MINIMUM LIMIT

Q309=0 ;PGM STOP TOLERANCE

Q330=0 ;TOOL

Q498=0 ;REVERSE TOOL

Q531=0 ;ANGLE OF INCIDENCE?

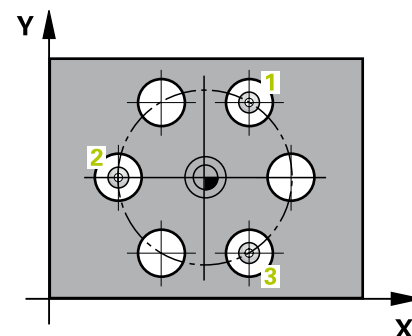
16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430)

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 467) 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

MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430) 16.12

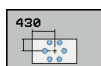
Please note while programming:



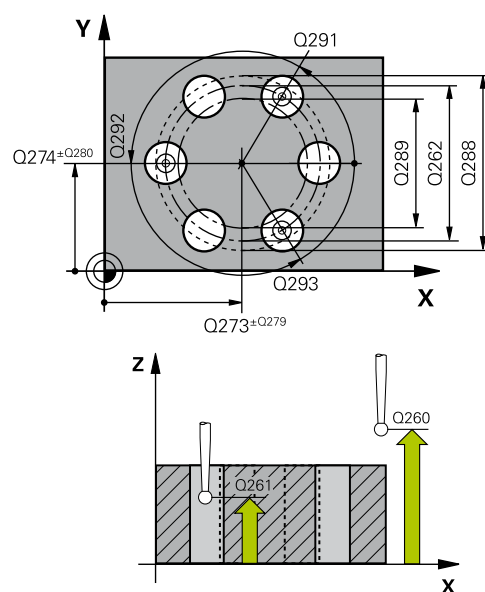
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the secondary axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range 0 to 99999.9999
- ▶ **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- ▶ **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- ▶ **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- ▶ **Q261 Measuring height in probe axis?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q288 Maximum limit of size?**: Maximum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- ▶ **Q289 Minimum limit of size?**: Minimum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor 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 LIMIT

Q289=79.9 ;MINIMUM LIMIT

Q279=0.15 ;TOLERANCE 1ST CENTER

Q280=0.15 ;TOLERANCE 2ND CENTER

Q281=1 ;MEASURING LOG

16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430)

- ▶ **Q281 Measuring log (0/1/2)?**: 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.
- ▶ **Q309 PGM stop if tolerance exceeded?**: 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
- ▶ **Q330 Tool for monitoring?**: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 550). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
 - 0**: Monitoring inactive
 - >0**: Number or name of the tool that the TNC used for machining. You are able to apply a tool via soft key directly from the tool table.

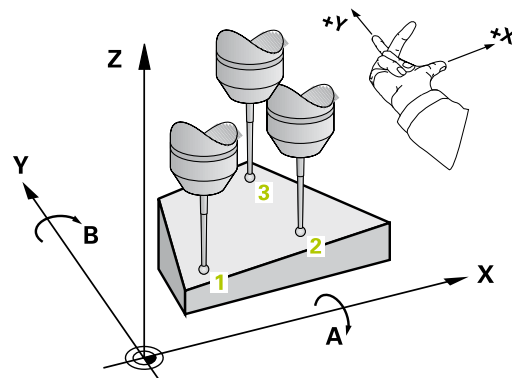
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

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 467) 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)

16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)

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.

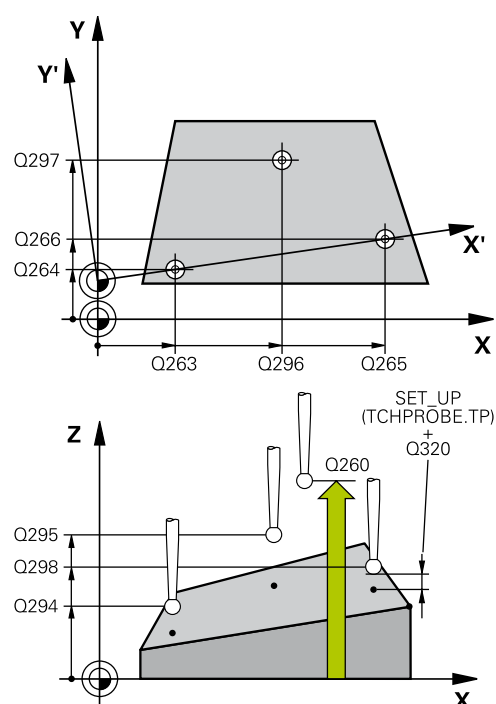
The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.

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



- ▶ **Q263 1st measuring point in 1st axis?** (absolute):
Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute):
Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q294 1st measuring point in 3rd axis?** (absolute):
Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute):
Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute):
Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q295 2nd measuring point in 3rd axis?** (absolute):
Coordinate of the second touch point in the touch probe axis. Input range -99999.9999 to 99999.9999



MEASURE PLANE (Cycle 431, DIN/ISO: G431) 16.13

- ▶ **Q296 3rd measuring point in 1st axis?** (absolute):
Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q297 3rd measuring point in 2nd axis?** (absolute):
Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q298 3rd measuring point in 3rd axis?** (absolute):
Coordinate of the third touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: 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 PNT IN 1ST AXIS
Q266=+80	;2ND PNT IN 2ND AXIS
Q295=+0	;2ND PNT IN 3RD AXIS
Q296=+90	;3RD PNT IN 1ST AXIS
Q297=+35	;THIRD POINT 2ND AXIS
Q298=+12	;3RD PNT IN 3RD AXIS
Q320=0	;SET-UP CLEARANCE
Q260=+5	;CLEARANCE HEIGHT
Q281=1	;MEASURING LOG

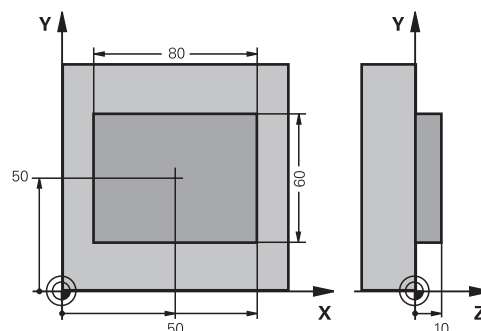
16.14 Programming Examples

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



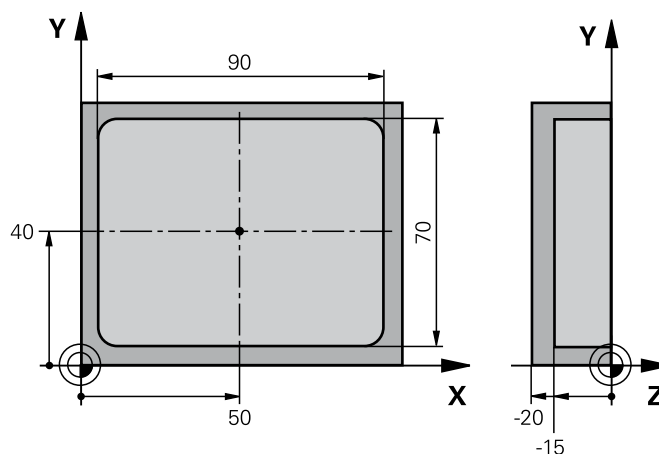
0 BEGIN PGM BEAMS MM		
1 TOOL CALL 69 Z		Tool call for roughing
2 L Z+100 R0 FMAX		Retract the tool
3 FN 0: Q1 = +81		Length of rectangle in X (roughing dimension)
4 FN 0: Q2 = +61		Length of rectangle in Y (roughing dimension)
5 CALL LBL 1		Call subprogram for machining
6 L Z+100 R0 FMAX		Retract the 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 TOLERANCE	Do not output an error message
Q330=0	;TOOL	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

Programming Examples 16.14

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 MILLNG	
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 ;2ND 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	

16.14 Programming Examples

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 TOLERANCE	Do not display an error message in case of a tolerance violation
Q330=0	;TOOL	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 Touch Probe Cycles: Special Functions

17.1 Fundamentals

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:

Soft key	Cycle	Page
	3 MEASURING Cycle for defining OEM cycles	593

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) touch-probe data, 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.

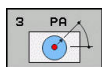
If the TNC was not able to determine a valid touch point, the program is run without error message. In this case the TNC assigns the value -1 to the 4th result parameter so that you can deal with the error yourself.

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.

17.2 MEASURE (Cycle 3)

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 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=ACT/1=REF):** Define whether the probing direction and measuring result should reference the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):
 - 0:** Probe in the current system and save the measuring result to the **ACT** 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 ABST +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. You can also define the distance by which the touch probe retracts 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:



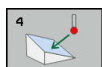
The TNC retracts the touch probe by no more than the retraction distance **MB** and does not pass the starting point of the measurement. This rules out any collision during retraction.

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.

17.3 MEASURING IN 3-D (Cycle 4)

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 range?:** 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 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=ACT/1=REF):** Specify whether the result of probing is to be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**):
 - 0:** Save measuring result to the **ACT** system
 - 1:** Save measuring result to 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 ABST+45 F100 MB50 REFERENCE SYSTEM:0

17.4 3D PROBING (Cycle 444)

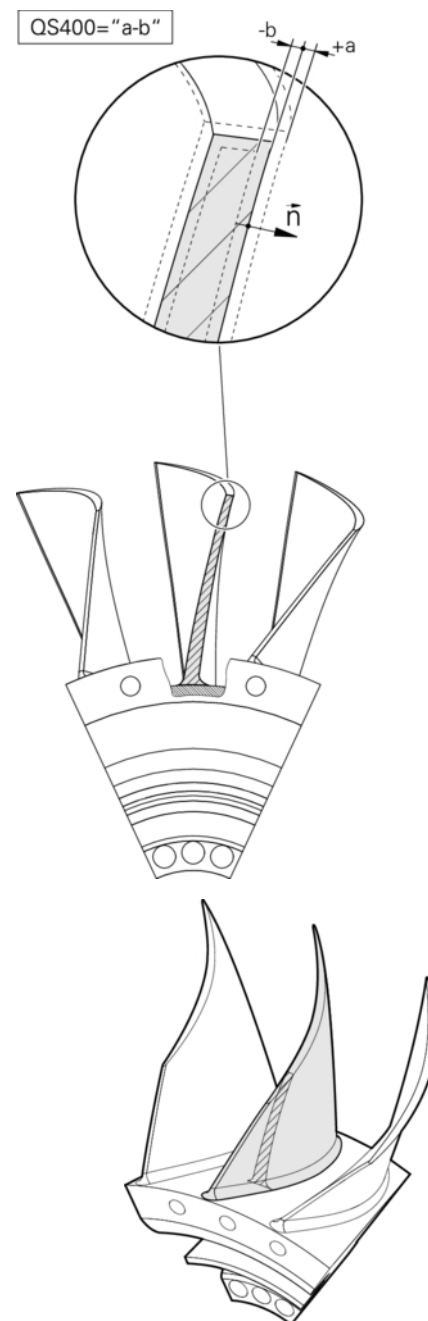
Cycle run

Cycle 444 checks one specific point on the surface of a component. This cycle is used, for example, to measure free-form surfaces of molded parts. It can be determined whether a point on the surface of the component lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle 444 probes any point in three dimensions, and determines the deviation to a nominal coordinate. A normal vector, defined in parameters Q581, Q582, and Q583, is used for this. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface, and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range QS400 defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the TNC outputs a log and the deviations are stored in the system parameters listed below.

Cycle run

- 1 Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + SET_UP value from tchprobe.tp table (TNC:\table\tchprobe.tp) + Q320. Pre-positioning takes a set-up clearance into account. For more information on the probing logic, see "Executing touch probe cycles", page 467
- 2 The touch probe then approaches the nominal coordinate. The probing path is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
- 3 After the TNC has saved the position, the touch probe is retracted and stopped. The TNC saves the measured coordinates of the contact point in Q parameters.
- 4 Finally, the TNC moves the touch probe back by that value against the probing direction that you defined in the parameter **MB**.



17.4 3D PROBING (Cycle 444)

System parameter

The TNC stores the results of the probing process in the following parameters:

System parameter	Meaning
Q151	Measured position in reference axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in reference axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3-D deviation <ul style="list-style-type: none"> ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap

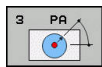
Log function

Once probing has finished, the TNC generates a log in HTML format. The TNC saves the log in the same folder in which the *.h file is located (as long as no path is configured for FN16).

The log includes the following contents:

- Defined nominal coordinate
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")
- (If a tolerance QS400 was defined) Upper and lower allowances are output, as well as the determined deviation along the normal vector
- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path

Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- ▶ **Q581 Surface-normal in ref. axis?** Enter here the surface normal in the main axis direction. The surface normal of a point is normally output with the aid of a CAD/CAM system. Input range: -10 to 10
- ▶ **Q582 Surface-normal in minor axis?** Enter here the surface normal in the minor axis direction. The surface normal of a point is normally output with the aid of a CAD/CAM system. Input range: -10 to 10
- ▶ **Q583 Surface-normal in tool axis?** Enter here the surface normal in the tool axis direction. The surface normal of a point is normally output with the aid of a CAD/CAM system. Input range: -10 to 10
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999

NC blocks

4 TCH PROBE 444 PROBING IN 3-D	
Q263=+0	;1ST POINT 1ST AXIS
Q264=+0	;1ST POINT 2ND AXIS
Q294=+0	;1ST POINT 3RD AXIS
Q581=+1	;NORMAL IN REF. AXIS
Q582=+0	;NORMAL IN MINOR AXIS
Q583=+0	;NORMAL IN TOOL AXIS
Q320=+0	;SET-UP CLEARANCE?
Q260=100	;CLEARANCE HEIGHT?
QS400="1-1"	;TOLERANCE
Q309=+0	;ERROR REACTION

17.4 3D PROBING (Cycle 444)

- **QS400 Tolerance value?** Enter here a tolerance range that is monitored by the cycle. The tolerance defines the deviation permitted along the surface normals. This deviation is determined between the nominal coordinate and the actual coordinate of the component. (The surface normal is defined by Q581 through Q583, and the nominal coordinate is defined by Q263, Q264, and Q294) The tolerance value is divided over the axes, depending on the normal vector:

Example: QS400 ="0.4-0.1" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate -0.1. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate - 0.1".

Example: QS400 ="0.4" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate".

Example: QS400 ="-0.1" means: upper allowance = nominal coordinate, lower allowance = nominal coordinate -0.1. The following tolerance range thus results for the cycle: "nominal coordinate" to "nominal coordinate - 0.1".

Example: QS400 =" " means: No tolerance band.

Example: QS400 ="0" means: No tolerance band.

Example: QS400 ="0.1+0.1" means: No tolerance band.

- **Q309 Reaction to tolerance error?** Specify whether the TNC is to interrupt program run and output a message if a deviation is detected:
 - 0:** If the tolerance is exceeded, do not interrupt program run, do not output an error message
 - 1:** If the tolerance is exceeded, interrupt program run and output an error message
 - 2:** If the determined actual coordinate along the surface normal vector is less than the nominal coordinate, the TNC outputs a message and interrupts program run. An undersize has occurred. On the other hand, there is no error reaction if the value determined along the surface normal vector is greater than the nominal coordinate.

Please note while programming:

In order to achieve exact results from the touch probe being used, a 3-D calibration should be conducted before probing with Cycle 444. Software option 92 3D-ToolComp is required for 3-D calibration.

Cycle 444 generates a measuring log in HTML format.

An error message is output if a mirroring (Cycle 8) or scaling (Cycle 11, 26) is active before Cycle 444 is run.

Depending on the setting of the parameter CfgPresetSettings, it is checked during probing whether the positions of the rotary axes match the tilting angles (3-D ROT). The TNC displays an error message if that is not the case.

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.

Cycle 444 references all coordinates to the input system.


The TNC writes the measured values to return parameters (see "Cycle run", page 597).

The workpiece status good/rework/scrap is set via Q parameter Q^83, independent of parameter Q309 (see "Cycle run", page 597).

17.5 Calibrating a touch trigger probe

17.5 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 the **CALIBRATE TS** soft key
 - ▶ Select the calibration cycle

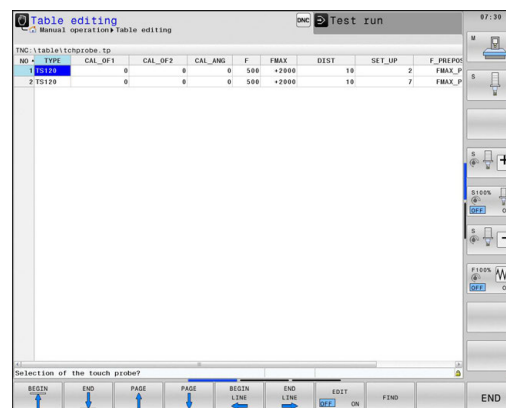
Calibration cycles of the TNC

Soft key	Function	Page
	Calibrating the length	608
	Measure the radius and the center offset using a calibration ring	610
	Measure the radius and the center offset using a stud or a calibration pin	612
	Measure the radius and the center offset using a calibration sphere	604

17.6 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: \ *.



Please make sure the correct tool number is active when you use the touch probe system. Regardless of whether you want to use a touch probe cycle in automatic mode or **Manual operation** mode.



For more information, see Chapter Touch probe table

17.7 CALIBRATE TS (Cycle 460, DIN/ISO: G460)

17.7 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.

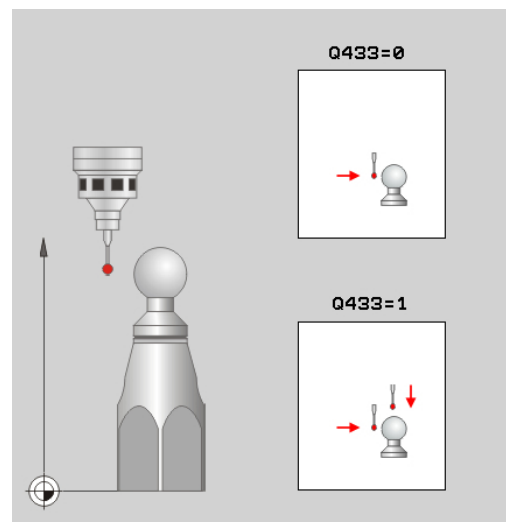
It is also possible to capture 3-D calibration data. Software option 92, 3D-ToolComp, is required for this purpose. 3-D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3-D calibration data are stored under TNC:\Table \CAL_TS<T-Nr.>_<T-Idx.>.3DTC. The DR2TABLE column of the tool table refers to the 3DTC table. The 3-D calibration data are then taken into account when probing. This 3-D calibration is necessary if you want to achieve a very high accuracy with Cycle 444 3-D Probing (see "3D PROBING (Cycle 444)", page 597).

Cycle run

The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

Radius calibration Q433=0

- 1 Clamp the calibration sphere. Ensure the prevention of 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 TNC first moves in the plane, depending on the reference angle (Q380).
- 4 The TNC then positions the touch probe in touch-probe axis.
- 5 The probing process starts, and the TNC begins by searching for the equator of the calibration sphere
- 6 Once the equator has been determined, the radius calibration begins
- 7 Finally, the TNC returns the touch probe in the touch-probe axis to the height at which it had been pre-positioned.



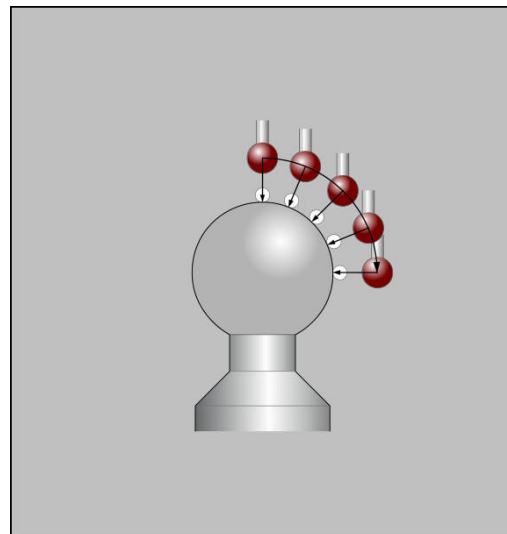
Radius and length calibration Q433=1

- 1 Clamp the calibration sphere. Ensure the prevention of 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 TNC first moves in the plane, depending on the reference angle (Q380).
- 4 The TNC then positions the touch probe in touch-probe axis.
- 5 The probing process starts, and the TNC begins by searching for the equator of the calibration sphere
- 6 Once the equator has been determined, the radius calibration begins
- 7 Then the TNC returns the touch probe in the touch-probe axis to the height at which it had been pre-positioned.
- 8 The TNC ascertains the length of the touch probe at the north pole of the calibration sphere
- 9 At the end of the cycle the TNC returns the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

The setting in parameter **Q455** specifies whether you can perform an additional 3-D calibration.

3-D calibration Q455= 1...30

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 After calibration of the radius or length, the TNC retracts the touch probe in touch-probe axis. Then the TNC positions the touch probe over the north pole
- 3 The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are thus determined
- 4 You can specify the number of probing points between the north pole and the equator. This number depends on input parameter Q455. A value between 1 and 30 can be programmed. If you program Q455=0, no 3-D calibration will be performed.
- 5 The deviations determined during the calibration are stored in a 3DTC table.
- 6 At the end of the cycle the TNC returns the touch probe in the touch-probe axis to the height at which it had been pre-positioned.



17.7 CALIBRATE TS (Cycle 460, DIN/ISO: G460)

Please note while programming:



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



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 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 program a tool call to define the touch-probe axis.

Pre-position the touch probe so that it is located approximately above the center of the calibration sphere.

If you program Q455=0, the TNC will not perform a 3-D calibration.

If you program Q455=1-30, there will be a 3-D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles. If you use Cycle 444, you should first perform a 3-D calibration.

If you program Q455=1-30, a table will be stored under TNC:\Table\CAL_TS<T-NR.>_<T-Idx.>.3DTC. <T-NR> is the number of the touch probe, and <Idx> is its index.

If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.

If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.



- ▶ **Q407 Radius of calib. sphere?:** Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q301 Move to clearance height (0/1)?:** 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
- ▶ **Q423 NUMBER OF PROBES?** (absolute): Number of measuring points on the diameter. Input range 0 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (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
- ▶ **Q433 Calibrate length (0/1)?:** Define whether the TNC is to calibrate the touch probe length after radius calibration, as well:
0: Do not calibrate touch probe length
1: Calibrate touch probe length
- ▶ **Q434 Datum for length?** (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
- ▶ **Q455 No. of points for 3-D calibrtn.?** Enter the number of probing points for 3-D calibration. A value of about 15 probing points is useful. If 0 is entered here, no 3-D calibration is performed. During 3-D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3-D calibration. Input range: 1 to 30

NC blocks

5 TCH PROBE 460 CALIBRATION OF TS ON A SPHERE	
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	;PRESET
Q455=15	;NO. POINTS 3-D CAL.

17.8 CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461)

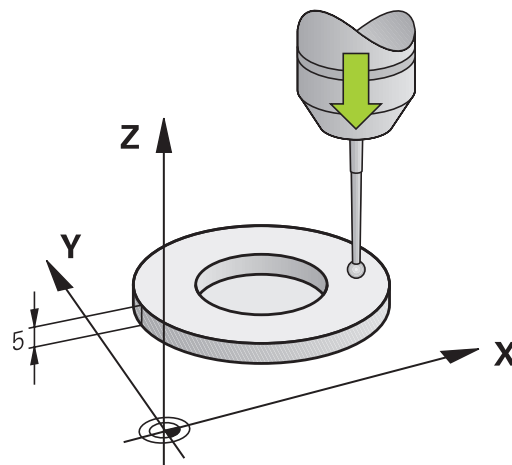
17.8 CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461)

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.



CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461) 17.8

Please note while programming:



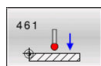
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



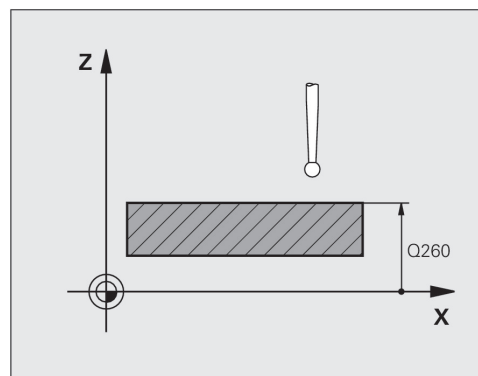
The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

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.



- **Q434 Datum for length?** (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 TS CALIBRATION OF
TOOL LENGTH

Q434=+5 ;PRESET

17.9 CALIBRATE TS RADIUS INSIDE (Cycle 462, DIN/ISO: G462)**17.9 CALIBRATE TS RADIUS INSIDE
(Cycle 462, DIN/ISO: G462)****Cycle run**

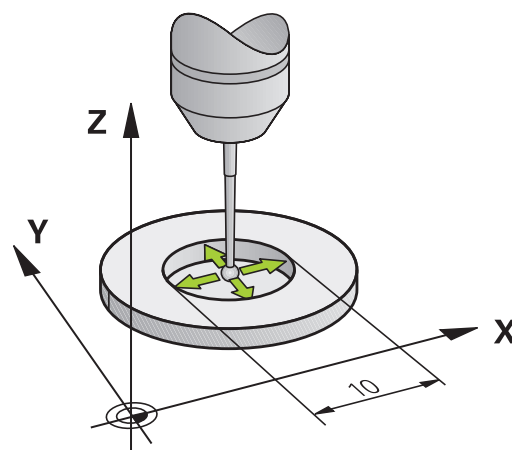
Before starting the calibration cycle, you need to preposition the touch probe in the center of the calibration ring and at the required measuring height.

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."



CALIBRATE TS RADIUS INSIDE (Cycle 462, DIN/ISO: G462) 17.9

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.

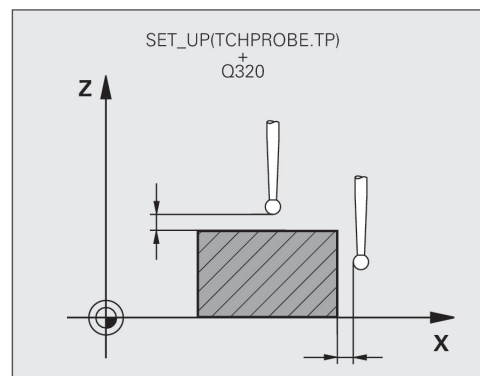


In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information.

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.



- ▶ **Q407 Radius of calibr. stud?**: Diameter of the ring gauge. Input range 0 to 99.9999
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q423 NUMBER OF PROBES?** (absolute): Number of measuring points on the diameter. Input range 0 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (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 CALIBRATION OF A TS IN A RING

Q407=+5 ;RING RADIUS

Q320=+0 ;SET-UP CLEARANCE

Q423=+8 ;NO. OF PROBE POINTS

Q380=+0 ;REFERENCE ANGLE

17.10 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, DIN/ISO: G463)**17.10 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 safety 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."

CALIBRATE TS RADIUS OUTSIDE (Cycle 463, DIN/ISO: G463) 17.10

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.

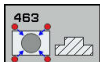
The center offset can be determined only with a suitable touch probe.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

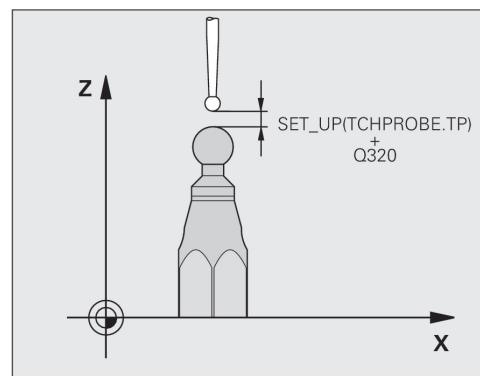


In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information.

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.



- ▶ **Q407 Radius of calibr. stud?:** Diameter of the ring gauge. Input range 0 to 99.9999
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q301 Move to clearance height (0/1)?:** 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
- ▶ **Q423 NUMBER OF PROBES?** (absolute): Number of measuring points on the diameter. Input range 0 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (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 CALIBRATION 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

**Visual Setup
Control VSC
(software option
136)**

Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Fundamentals

In order to use the Visual Setup Control you require the following components:

- Software: Option 136 Visual Setup Control (VSC)
- Hardware: Camera system of HEIDENHAIN

Application



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

Camera-based checking of the setup situation (option number 136 Visual Setup Control) allows you to monitor the current setup situation before and during processing, and to compare this with a safe target status. After setup, simple cycles for automatic monitoring are available.

Reference images of the current working space are recorded in the camera system. With Cycles 600 **GLOBAL WORKING SPACE** or 601 **LOCAL WORKING SPACE**, the TNC produces an image of the working space and compares the image with previously prepared reference images. These cycles can highlight irregularities in the working space. The operator decides whether the NC program is interrupted in the event of an error or continues to run.

Using VSC offers the following advantages:

- The control can recognize elements (e.g. tools, fixtures, etc.) that are in the working space once the program has started
- If you always want to clamp a workpiece at the same position (e.g. hole at top right), the control can check the clamping situation
- For documentation purposes you can generate an image of the current workspace (e.g. of a clamping situation that is rarely used)

Terms

The following terms are used in connection with VSC:

Term	Explanation
Reference image	A reference image shows a situation in the working space that you regard as safe. Therefore only create reference images showing safe, non-hazardous situations.
Mid-value image	The control creates a mid-value image taking into account all reference images. The control compares new images with the mid-value image as part of evaluation.

Camera-based monitoring of the setup situation VSC (option number 136) 18.1

Term	Explanation
Error	<p>If you generate an image showing a poor situation (such as an incorrectly clamped workpiece), you can generate an "image of error"</p> <p>It is not advisable to highlight an error image as a reference image.</p>
Monitoring area	<p>Denotes an area that you highlight with the mouse. When evaluating new images, the control only refers to this area. Parts of images outside the monitoring area have no effect on the results of the monitoring process. Several monitoring areas can be defined. Monitoring areas are not linked to images.</p>
Error	<p>Area on an image containing a deviation from the desired position. Errors always refer to the image for which they were saved (image of error) or to the image most recently evaluated.</p>
Monitoring phase	<p>No further reference images are produced in the monitoring phase. You can use the cycle for automatic monitoring of your working space. In this phase, the control only issues a warning if it finds a deviation when comparing images.</p>

Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number136)

Produce live image



In the **Manual operation** mode, you can display and save the current camera view as a live image.

The control does not use the image captured here for automatic checking of the clamping situation. Images produced in this menu may be used for documentation and traceability. For example, you could record the current setup situation. The control saves the image produced as a .png file in **TNC:\system\visontool\live_view**. The name of the saved image is made up of the date and time it was created.







Procedure

Proceed as follows to save the camera’s live image:

-  ► Press the **CAMERA** soft key
-  ► Press the **LIVE IMAGE** soft key: The TNC shows you the current camera view
-  ► Press the **SAVE IMAGE** soft key: Create a live image from the current camera view

Options in Live Image mode

The control provides the following options:

Soft key	Function
	Increase camera brightness The settings made here only affect Live Image mode, and have no influence on pictures taken in automatic mode.
	Reduce camera brightness The settings made here only affect Live Image mode, and have no influence on pictures taken in automatic mode.
	Configuring the field of view of the camera Refer to your machine manual. These settings can only be made after entering a code number.
	Go back to the previous screen

Camera-based monitoring of the setup situation VSC (option number 136) 18.1

Manage monitoring data

In the **Manual operation** mode you can manage images from Cycles 600 and 601.

Proceed as follows to enter the monitoring data:



- ▶ Press the **CAMERA** soft key



- ▶ Press the **MONITORING DATA MANAGEMENT** soft key: The control shows a list of the NC programs monitored

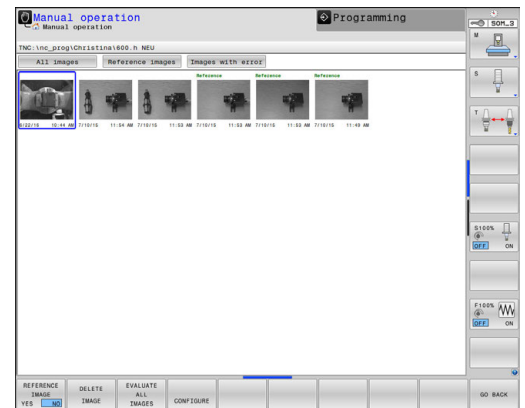


- ▶ Press the **OPEN** soft key: The control shows a list of monitoring points
- ▶ Edit the desired data

Select data

You can select the buttons with the mouse. These interfaces make it easier to search and show results in a manageable way.

- **All images:** Display all images for this monitoring file
- **Reference images:** Only display reference images
- **Images with error:** Display all images where you have highlighted an error



Features of the monitoring data management

Soft key	Function
	Mark selected image as a reference image Please note: A reference image shows a situation in the working space that you regard as safe. All reference images are used as part of the evaluation process. If you add or remove an image as a reference image, this has an effect on the results of image evaluation.
	Delete image currently selected
	Carry out automatic image evaluation The control carries out an image evaluation according to the reference images and the monitoring areas.
	Change monitoring area or highlight an error Further Information: "Configuration", page
	Go back to the previous screen If you change the configuration, the control carries out an image evaluation.

Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Overview

The TNC provides two cycles you can use for visual setup control in the **Programming** mode of operation:



- The soft-key row shows all available touch probe functions divided into groups.



- Press the **MONITORING WITH CAMERA** soft key

Soft key	Cycle	Page
	600 GLOBAL WORKING SPACE	626
	601 LOCAL WORKING SPACE	631

Camera-based monitoring of the setup situation VSC (option number136) 18.1

Results of the image evaluation

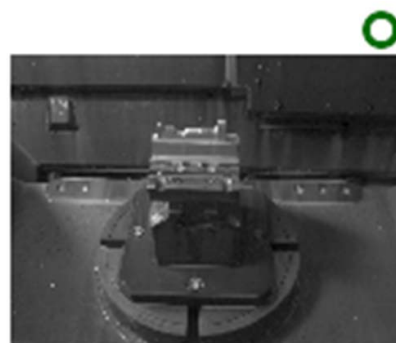
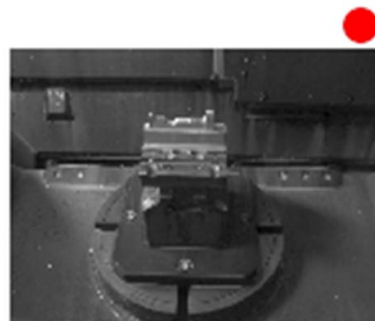
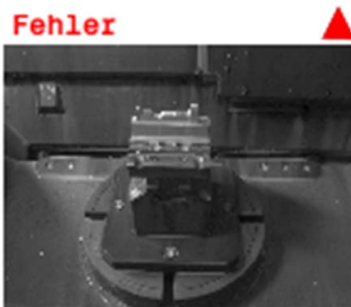
The results of the image evaluation depend on the monitoring area and the reference images. When evaluating all images, each image is evaluated according to the current configuration and the results are compared with the data last saved.

If you change the monitoring area or add/delete reference images, the images may be tagged with the following symbol:

- **Triangle:** You have changed the monitoring data, e.g. tagged an image with errors as a reference image or deleted a monitoring area. This makes the monitoring less sensitive.

This has an effect on your reference images and on the mid-value image. As a result of your change to the configuration, the control can no longer detect errors that had been saved for this image! If you would like to continue, confirm the reduced monitoring sensitivity and the new settings will be accepted.

- **Solid circle:** You have changed the monitoring data, monitoring is more sensitive.
- **Blank circle:** No error message: All deviations saved in the image have been recognized, monitoring has not identified any conflicts.

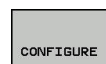


Visual Setup Control VSC (software option 136)

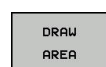
18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Configuration

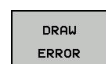
You are now able to change your settings regarding the monitoring area and error area at any time. When you press the **CONFIGURE** soft key, the soft key display changes and you can change your settings.



- You also are now able to edit your previously entered settings. If you carry out the change in this menu, the result of the image evaluation may change. The same monitoring area applies for all reference images. (For more information, see "Results of the image evaluation", page 621.)



- You are able to click on the image and draw up a rectangular frame. This way you define the monitoring area. (For more information, see "Fundamentals", page 616.) If you define monitoring areas in a setting that is always exposed or in which differences in contrast are to be expected, false alerts will be displayed. If you draw a new monitoring area or change or delete an already drawn up monitoring area, this will affect the result of the image evaluation. Due to the changed settings the TNC must check whether these changes may have an impact on the previously generated images.



- You are able to click on the image and draw up a rectangular frame. This way you define a new area with errors. The area is marked red. It is recommended to mark only errors that may reoccur exactly at this spot. It is not advisable to mark areas that are contaminated with chips or drilling fluid. The errors need to be reproducible in the exact same way. (For more information, see "Fundamentals", page 616.) If you define monitoring areas in a setting that is always exposed or in which differences in contrast are to be expected, false alerts will be displayed. If you draw a new area with errors or change or delete an already drawn up area of errors, this will affect the result of the image evaluation. Due to the changed settings the TNC must check whether these changes may have an impact on the previously generated images. You can also draw several areas with errors. It does not make sense to indicate errors on reference images.

Camera-based monitoring of the setup situation VSC (option number 136) 18.1

EVALUATE
IMAGE

- ▶ The TNC checks if or how the new settings affect this image: (For more information, see "Results of the image evaluation", page 621.)

EVALUATE
ALL
IMAGES

- ▶ The TNC checks if or how the new settings affect all images: (For more information, see "Results of the image evaluation", page 621.)

SAVE
AND
GO BACK

- ▶ Save current image and return to previous screen. If you changed the configuration, the TNC will perform an image evaluation. (For more information, see "Results of the image evaluation", page 621.)

GO BACK

- ▶ You discard all changes and return to the previously displayed screen.

Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Defining the monitoring area

The monitoring area is defined with the modes of operation Single Block or Block Scan. The TNC will prompt you to define a monitoring area. The TNC will display this prompt on the screen after you have started the cycle for the first time in the modes of operation Single Block or Block Scan.

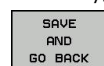
A monitoring area consists of one or more windows that you draw with your mouse. The TNC will only scan these areas of the image. Errors outside of the monitoring area will not be detected. The monitoring area is not linked to images but only with the appropriate monitoring file QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Monitoring areas may overlap.

Definition of the monitoring area:

- 1 Click with the mouse on the image and draw a new area
- 2 If you would like to define more than one window, press the **DRAW AREA** soft key, and repeat this process at the appropriate location

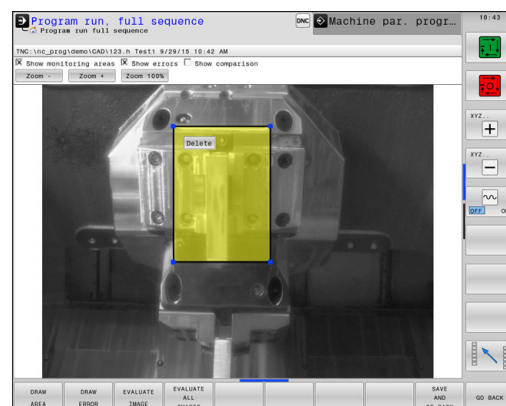
After you have defined the monitoring area you press the following soft key, for example:



- Save current image and return to previous screen.

This message is displayed: **Monitoring point is configured: Select the soft key.**

The status display at the top right of the image will display information about the minimum number of reference images, about the current number of reference images and the current number of error images.



Camera-based monitoring of the setup situation VSC (option number 136) 18.1

Possible queries

The VSC cycles enter a value in parameter Q601.

The following values are possible:

- Q601 = 1: No error
- Q601 = 2: Error
- Q601 = 3: You have yet not defined a monitoring area or you did not save enough reference images
- Q601 = 10: Internal error (no signal, faulty camera, etc.)

You can use parameter Q601 for internal queries.



More information about If-Then Decisions with Q parameter functions are found in the user manuals of the 'TNC 640, section 9.6

Here, a possible programming example for a query:

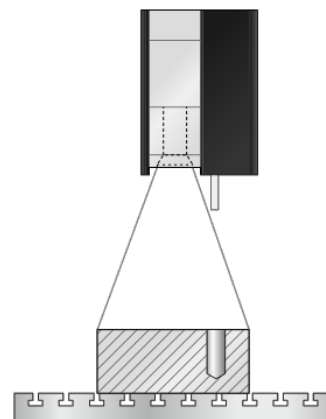
0 BEGIN PGM 5MM	
1 BLK FORM CYLINDER Z R42 L150	Blank form Cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TCH PROBE 601 LOCAL WORKING SPACE	Define Cycle 600
Q\$600 = OS ;MONITORING POINT	
Q309=+0 ;PGM STOP TOLERANCE	
Q613 = +0 ;KEEP CAMERA OPEN	
Q617 = 10 ;REFERENCE IMAGES	
4 FN 9: IF Q601 EQU 1 GOTO LBL 20	If parameter Q601 = 1, then jump to LBL 20
5 FN 9: IF Q601 EQU 2 GOTO LBL 21	If parameter Q601 = 2, then jump to LBL 21
6 FN 9: IF Q601 EQU 3 GOTO LBL 22	If parameter Q601 = 3, then jump to LBL 22
7 FN 9: IF Q601 EQU 10 GOTO LBL 23	If parameter Q601 = 10, then jump to LBL 23
8 TOOL CALL "GEAR_HOB_D75"	Call the tool
9 L X+... Y+... R0 FMAX	Program the machining
...	
...	
...	
57 LBL 21	Definition LBL 21
58 STOP	Program stop, the operator can check the condition of the workspace
59 LBL 0	
60 END PGM 5MM	

18.2 Workspace Global (Cycle 600)

18.2 Workspace Global (Cycle 600)

Application

With Cycle 600, Workspace Global, you monitor the workspace of your tooling machine. The TNC will generate an image of the current workspace from a position determined by your machine tool builder. Then, the TNC will match the image with previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 600 becomes effective as soon as it is defined, and does not have to be called. Before you can use the camera monitoring, you must first generate reference images (for further information, see "Creating reference images", page 626) and define a monitoring area (for further information, see "Monitoring phase", page 629).



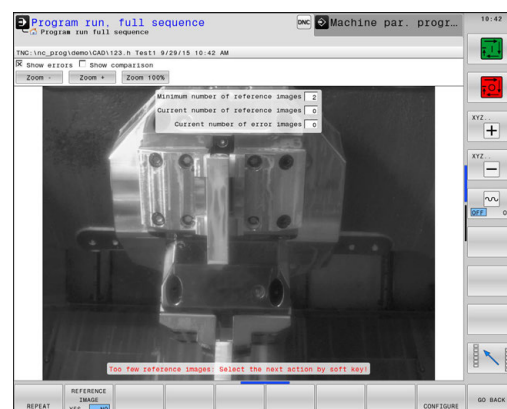
Creating reference images

The TNC will start generating reference images as soon as you run the cycle for the first time in the program modes Single Block or Block Scan.






The following cycle run applies as long as the TNC has not stored sufficient reference images. Specify the number of reference images with parameter Q617.

Cycle run

- 1 The camera will be mounted by the machine tool builder onto the main spindle.
- 2 The TNC automatically opens the camera lid.
- 3 The TNC will generate an image of the current condition and display it on the screen.
- 4 The first time this cycle is run, the following message appears at the bottom of the screen: **"Monitoring point not configured: Draw areas!"**
- 5 Define the monitoring area. (For more information, see "Defining the monitoring area", page 624)
- 6 You can decide whether the current image should be saved as a reference image or an error image, but you can also change the monitoring area. (For more information, see "Configuration").
- 7 Press the **GO BACK** soft key.
- 8 To complete the operation the TNC will close the camera lid.
- 9 Press the NC start and run your program as usual.



After you have defined the monitoring area, you can press the following soft keys in addition to the **GO BACK** soft key:

- | | |
|---|--|
|  | <ul style="list-style-type: none"> ▶ The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (See "Results of the image evaluation") |
|  | <ul style="list-style-type: none"> ▶ The status display at the top right displays the word "Reference". You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the soft key IMAGE OF ERROR turns gray. (For more information, see "Fundamentals", page 616) |
|  | <ul style="list-style-type: none"> ▶ The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an error image can never be a reference image at the same time, the soft key REFERENCE IMAGE (For more information, see "Fundamentals", page 616) |
|  | <ul style="list-style-type: none"> ▶ The soft key row changes. You are now able to change your previously entered settings with respect to the monitoring area and the sensitivity. If you carry out the change in this menu, this may impact all of your images. (For more information, see "Configuration", page 622) |
|  | <ul style="list-style-type: none"> ▶ The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (For more information, "Results of the image evaluation") |



As soon as the TNC generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected the following message is displayed: **Too few reference images: Select the next action by soft key!** This message is no longer displayed once the number of reference images defined number in parameter Q617 has been reached.



The TNC will generate mean image taking into account all reference images. During evaluation, new images are compared with the mean image under consideration of the variance. Only once all reference images are available does the cycle no longer stop if there are not enough reference images.

Visual Setup Control VSC (software option 136)

18.2 Workspace Global (Cycle 600)

Defining the monitoring area

The monitoring area is defined with the modes of operation Single Block or Block Scan. The TNC will prompt you to define a monitoring area. The TNC will display this prompt on the screen after you have started the cycle for the first time in the modes of operation Single Block or Block Scan.

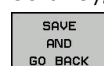
A monitoring area consists of one or more windows that you draw with your mouse. The TNC will only scan these areas of the image. Errors outside of the monitoring area will not be detected. The monitoring area is not linked to images but only with the appropriate monitoring file QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Monitoring areas may overlap.

Definition of the monitoring area:

- 1 Click with the mouse on the image and draw a new area
- 2 If you would like to define more than one window, press the **DRAW AREA** soft key, and repeat this process at the appropriate location

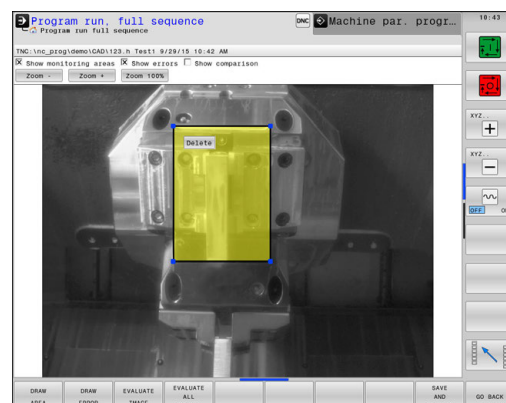
After you have defined the monitoring area you press the following soft key, for example:



- Save current image and return to previous screen.

This message is displayed: **Monitoring point is configured: Select the soft key.**

The status display at the top right of the image will display information about the minimum number of reference images, about the current number of reference images and the current number of error images.



Monitoring phase

Cycle run: Monitoring phase

- 1 The camera will be mounted by the machine tool builder onto the main spindle. The main spindle moves to a position defined by the machine tool builder.
- 2 After the TNC has reached this position, it will automatically open the camera lid.
- 3 The TNC will generate an image of the current condition.
- 4 There is then an image comparison between the mean image and the variance image (for more information, see "Fundamentals", page 616).
- 5 Depending on whether the TNC detects an "error" (deviation), the TNC can now force a program cancellation (for further information, see "Fundamentals", page 616). If parameter Q309=1 is set, the TNC will display the image on the screen upon detecting an error. If parameter Q309=0 is set, no image will be displayed on the screen and the program will not be stopped.
- 6 To complete the operation the TNC will close the camera lid.

Please note while programming:



In addition to the attribute Reference image, you can also assign the attribute Error image to your images. This assignment may impact the image evaluation.

Remember the following:

- ▶ A reference image may never be an error image at the same time..



If you change the monitoring area, this has an impact on all images.

- ▶ Ideally, you define the monitoring area only once at the beginning and then make no or only little changes to it.



The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.

- ▶ Specify a sensible number of reference images in parameter Q617. (Approx. value: 10 images).
- ▶ You can also create more reference images than you have specified in Q617..



Your machine must be prepared for camera-based monitoring!

18.2 Workspace Global (Cycle 600)

Danger of contaminating the camera due to open camera lid with parameter Q613.
This could lead to blurred images, the camera may be damaged.
Close the camera lid before continuing with the process.

Danger of collision during automatic positioning of the camera.
The camera and your machine may be damaged.
Ask your machine manufacturer to tell you the point that the camera pre-positions to. Your machine tool builder specifies the coordinates which Cycle 600 positions to.

Cycle parameters



- ▶ **QS600 (String-Parameter) Name of monitoring point?:** Enter the name of your monitoring file
- ▶ **Q616 Feed rate for positioning?:** Feed rate the TNC uses to position the camera. Here, TNC moves to a position defined by the machine tool builder.
- ▶ **Q309 PGM stop if tolerance exceeded?:** (0/1)
Specify whether the TNC stops the program if an error is detected.
0: Program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter Q601 is written to even if Q309=0.
1: Program will stop after detecting an error, the generated image will be displayed in the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the TNC has generated enough reference images. If an error is detected, the TNC will display a message.
- ▶ **Q617 Number of reference images?:** Number of reference images that are required by the TNC for monitoring.

NC blocks

4 TCH PROBE 600 GLOBAL WORKING SPACE

QS600="OS";MONITORING POINT

Q616=500 ;FEED RATE FOR POSITIONING

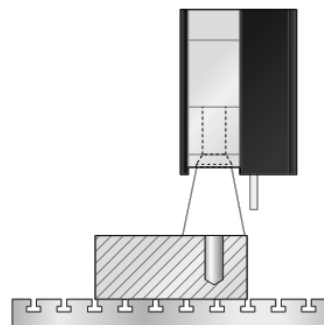
Q309=1 ;PGM STOP TOLERANCE

Q617=10 ;REFERENCE IMAGES

18.3 Workspace Local (Cycle 601)

Application

With Cycle 601, Workspace Local, you monitor the workspace of your tooling machine. The TNC will generate an image of the current workspace from the position of the spindle at the point in time of the cycle call. Then, the TNC will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 601 becomes effective as soon as it is defined, and does not have to be called. Before you can use the camera monitoring, you must first generate reference images (for further information, see "Creating reference images") and define a monitoring area (for further information, see "Monitoring phase", page 634).



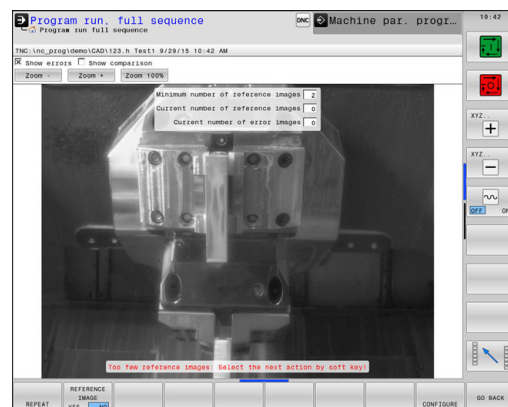
Creating reference images

The TNC will start generating reference images as soon as you run the cycle for the first time in the program modes Single Block or Block Scan.

The following cycle run applies as long as the TNC has not stored sufficient reference images. Specify the number of reference images with parameter Q617.






Cycle run

- 1 The camera will be mounted by the machine tool builder onto the main spindle.
- 2 The TNC automatically opens the camera lid.
- 3 The TNC will generate an image of the current condition and display it on the screen.
- 4 The first time this cycle is run, the following message appears at the bottom of the screen: **"Monitoring point not configured: Draw areas!"**
- 5 Define the monitoring area. (For more information, see "Defining the monitoring area", page 624)
- 6 You can decide whether the current image should be saved as a reference image or an error image, but you can also change the monitoring area. (For more information, see "Configuration").
- 7 Press the **GO BACK** soft key.
- 8 To complete the operation the TNC will close the camera lid.
- 9 Press the NC start and run your program as usual.



18.3 Workspace Local (Cycle 601)

After you have defined the monitoring area, you can press the following soft keys in addition to the **GO BACK** soft key:

- | | |
|---|--|
|  | <ul style="list-style-type: none"> ▶ The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (See "Results of the image evaluation") |
|  | <ul style="list-style-type: none"> ▶ The status display at the top right displays the word "Reference". You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the soft key IMAGE OF ERROR turns gray. (For more information, see "Fundamentals", page 616) |
|  | <ul style="list-style-type: none"> ▶ The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an error image can never be a reference image at the same time, the soft key REFERENCE IMAGE (For more information, see "Fundamentals", page 616) |
|  | <ul style="list-style-type: none"> ▶ The soft key row changes. You are now able to change your previously entered settings with respect to the monitoring area and the sensitivity. If you carry out the change in this menu, this may impact all of your images. (For more information, see "Configuration", page 622) |
|  | <ul style="list-style-type: none"> ▶ The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (For more information, "Results of the image evaluation") |



As soon as the TNC generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected the following message is displayed: **Too few reference images: Select the next action by soft key!** This message is no longer displayed once the number of reference images defined number in parameter Q617 has been reached.



The TNC will generate mean image taking into account all reference images. During evaluation, new images are compared with the mean image under consideration of the variance. Only once all reference images are available does the cycle no longer stop if there are not enough reference images.

Defining the monitoring area

The monitoring area is defined with the modes of operation Single Block or Block Scan. The TNC will prompt you to define a monitoring area. The TNC will display this prompt on the screen after you have started the cycle for the first time in the modes of operation Single Block or Block Scan.

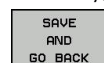
A monitoring area consists of one or more windows that you draw with your mouse. The TNC will only scan these areas of the image. Errors outside of the monitoring area will not be detected. The monitoring area is not linked to images but only with the appropriate monitoring file QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Monitoring areas may overlap.

Definition of the monitoring area:

- 1 Click with the mouse on the image and draw a new area
- 2 If you would like to define more than one window, press the **DRAW AREA** soft key, and repeat this process at the appropriate location

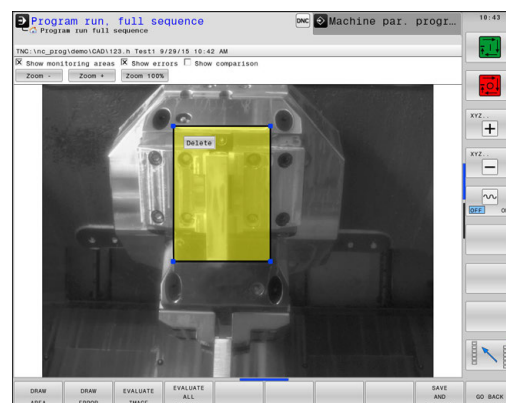
After you have defined the monitoring area you press the following soft key, for example:



- Save current image and return to previous screen.

This message is displayed: **Monitoring point is configured: Select the soft key.**

The status display at the top right of the image will display information about the minimum number of reference images, about the current number of reference images and the current number of error images.



18.3 Workspace Local (Cycle 601)**Monitoring phase**

The monitoring phase starts as soon as the TNC has generated enough reference images.

Cycle run: Monitoring phase

- 1 The camera will be mounted by the machine tool builder onto the main spindle.
- 2 The TNC automatically opens the camera lid.
- 3 The TNC will generate an image of the current condition.
- 4 There is then an image comparison between the mean image and the variance image (for more information, see "Fundamentals", page 616)
- 5 Depending on whether the TNC detects an "error" (deviation), the TNC can now force a program cancellation (for further information, "Results of the image evaluation"). If parameter Q309=1 is set, the TNC will display the image on the screen upon detecting an error. If parameter Q309=0 is set, no image will be displayed on the screen and the program will not be stopped.
- 6 Depending on parameter Q613, the TNC either leaves the camera lid open or closes it.

Please note while programming:

In addition to the attribute Reference image, you can also assign the attribute Error image to your images. This assignment may impact the image evaluation. Remember the following:

- ▶ A reference image may never be an error image at the same time..



If you change the monitoring area, this has an impact on all images.

- ▶ Ideally, you define the monitoring area only once at the beginning and then make no or only little changes to it.



The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.

- ▶ Specify a sensible number of reference images in parameter Q617. (Approx. value: 10 images)
- ▶ You can also create more reference images than you have specified in Q617.



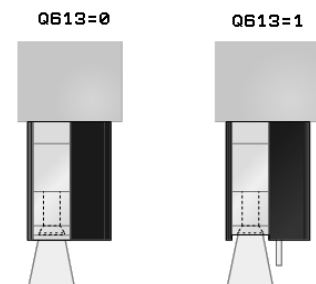
Your machine must be prepared for camera-based monitoring!

Danger of contaminating the camera due to open camera lid with parameter Q613.
This could lead to blurred images, the camera may be damaged.
Close the camera lid before continuing with the process!

Cycle parameters



- ▶ **QS600** (String-Parameter) **Name of monitoring point?:** Enter the name of your monitoring file
- ▶ **Q309 PGM stop if tolerance exceeded?:** (0/1)
Specify whether the TNC stops the program if an error is detected.
0: Program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter Q601 is written to even if Q309=0.
1: Program will stop after detecting an error, the generated image will be displayed in the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the TNC has generated enough reference images. If an error is detected, the TNC will display a message.
- ▶ **Q613 Keep camera cap open?:** (0/1) Specify whether the TNC should keep the camera cap open after monitoring.
0: The TNC closes the camera cap after running Cycle 601.
1: The TNC keeps camera cap open after running Cycle 601. This function is only recommended if you would like to generate yet another image of the workspace in a different position after the first cycle call of Cycle 601. In order to so, program the new position on a straight line block and then call up Cycle 601 with a new monitoring point. Program Q613=0 before continuing with the cutting operation!
- ▶ **Q617 Number of reference images?:** Number of reference images that are required by the TNC for monitoring.



NC blocks

4 TCH PROBE 601 WORKSPACE LOCAL

QS600="OS";MONITORING POINT

Q309=+1 ;PGM STOP IF ERROR

Q613=0 ;KEEP CAMERA OPEN

Q617=10 ;REFERENCE IMAGES

19

**Touch Probe
Cycles: Automatic
Kinematics
Measurement**

Touch Probe Cycles: Automatic Kinematics Measurement

19.1 Kinematics measurement with TS touch probes (KinematicsOpt option)

19.1 Kinematics measurement with TS touch probes (KinematicsOpt option)

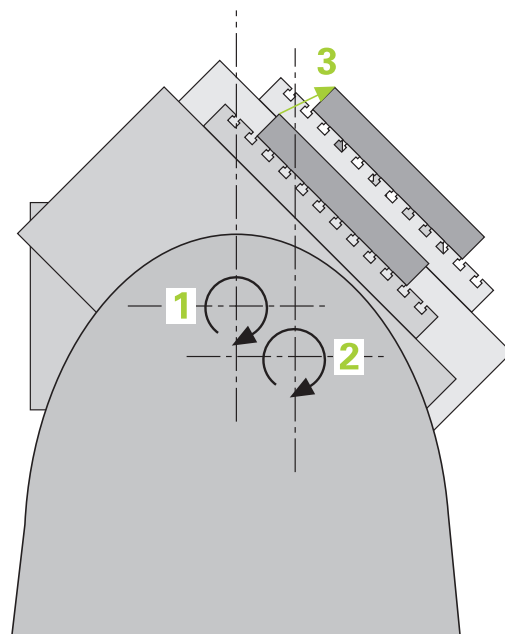
Fundamentals

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.



Kinematics measurement with TS touch probes 19.1 (KinematicsOpt option)

Overview

The TNC offers cycles that enable you to automatically save, check and optimize the machine kinematics:

Soft key	Cycle	Page
	450 SAVE KINEMATICS Automatic saving and restoring of kinematic configurations	641
	451 MEASURE KINEMATICS Automatic checking or optimizing of machine kinematics	644
	452 PRESET COMPENSATION Automatic checking or optimizing of machine kinematics	658

19.2 Prerequisites

19.2 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 defined the machine parameters for **CfgKinematicsOpt** in the configuration data. **maxModification** specifies the tolerance limit starting from which the TNC is to display a message if the changes to the kinematic data exceed this limit value. **maxDevCalBall** defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter. **mStrobeRotAxPos** defines an M function that is specifically configured by the machine manufacturer and is used to position the rotary axes.

Please note while programming:



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



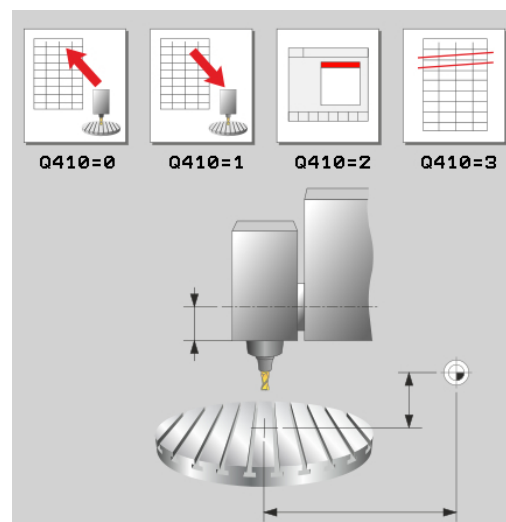
If an M function has been defined in machine parameter **mStrobeRotAxPos**, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450).

If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option)

Cycle run

With 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).

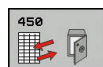
With the **Restore** mode, note that

- 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.

Touch Probe Cycles: Automatic Kinematics Measurement

19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option)

Cycle parameters



- ▶ **Q410 Mode (0/1/2/3)?**: 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.
- ▶ **Q409/QS409 Name of data record?**: Number or name of the data block designator. For numbers you can enter values from 0 to 99999, and when entering letters the string length may be no longer than 16 characters. 16 memory spaces in total are available. Q409 has no function if Mode 2 has been selected. Wildcards can be used for searches in Modes 1 and 3 (Restore and Delete). If the TNC finds several possible data records because of the wildcards, the TNC restores the mean values of the data (Mode 1) or deletes all data blocks after confirmation (Mode 3). You can use the following wildcards for searching:
 - ?**: 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

Q409=947 ;MEMORY DESIGNATION

Restoring data blocks

5 TCH PROBE 450 SAVE KINEMATICS

Q410=1 ;MODE

Q409=948 ;MEMORY DESIGNATION

Displaying all saved data blocks

5 TCH PROBE 450 SAVE KINEMATICS

Q410=2 ;MODE

Q409=949 ;MEMORY DESIGNATION

Deleting data blocks

5 TCH PROBE 450 SAVE KINEMATICS

Q410=3 ;MODE

Q409=950 ;MEMORY DESIGNATION

Logging function

After running Cycle 450, the TNC will create a measuring log (**TCHPRAUTO.HTML**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

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.

Make sure that you delete empty files with the name **TNC:\table\DATA450.KD**, if any, before starting Cycle 450. If there is an empty storage table (**TNC:\table\DATA450.KD**) that does not contain any lines yet, an error message will be displayed upon starting to run Cycle 450. In this case, delete the empty storage table and call the cycle again.

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).

Touch Probe Cycles: Automatic Kinematics Measurement

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

Cycle run

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:



MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option) 19.4

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

Touch Probe Cycles: Automatic Kinematics Measurement

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

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 = $(-90 - +90) / (4-1) = -60^\circ$
 - 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 = $(270^\circ - 90^\circ) / (4 - 1) = +60^\circ$
 - 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 coupling. 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 = $(90^\circ - -30^\circ) / (4 - 1) = 120^\circ / 3 = 40^\circ$

Measuring position 1 = $Q411 + 0 * \text{stepping angle} = -30^\circ \rightarrow -30^\circ$

Measuring position 2 = $Q411 + 1 * \text{stepping angle} = +10^\circ \rightarrow 9^\circ$

Measuring position 3 = $Q411 + 2 * \text{stepping angle} = +50^\circ \rightarrow 51^\circ$

Measuring position 4 = $Q411 + 3 * \text{stepping angle} = +90^\circ \rightarrow 90^\circ$

Touch Probe Cycles: Automatic Kinematics Measurement

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

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 657).

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.

Touch Probe Cycles: Automatic Kinematics Measurement

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

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 MP6602 is not equal to -1 (PLC macro positions the rotary axes), only start a measurement if 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.

The TNC ignores cycle definition data that applies to inactive axes.

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.

For angle optimization, the machine manufacturer must have adapted the configuration correspondingly. Angle optimization can be particularly useful on small, compact machines.

Angle compensation is only possible with option 52 **KinematicsComp**.



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.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option) 19.4

Cycle parameters



- ▶ **Q406 Mode (0/1/2):** Specify whether the TNC should check or optimize the active kinematics:
0: Check active machine 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 machine kinematics: The TNC measures the kinematics in the rotary axes you have defined. It then optimizes **the position of the rotary axes** of the active kinematics.
2: Optimize active machine kinematics: The TNC measures the kinematics in the rotary axes you have defined. It then optimizes **angle and position errors**. Software option 52 KinematicsComp is required for compensation of angle errors.
- ▶ **Q407 Radius of calib. sphere?:** Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- ▶ **Q320 Set-up clearance?** (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**
- ▶ **Q408 Retraction height?** (absolute): Input range 0.0001 to 99999.9999
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
>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.
- ▶ **Q253 Feed rate for pre-positioning?:** 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

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

- ▶ **Q380 Ref. angle in ref. axis?** (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
- ▶ **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q413 Angle of incidence in A axis?:** Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- ▶ **Q414 No. of meas. points in A (0...12)?:** 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
- ▶ **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q417 Angle of incidence in B axis?:** Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- ▶ **Q418 No. of meas. points in B (0...12)?:** 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
- ▶ **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q421 Angle of incidence in C axis?:** Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- ▶ **Q422 No. of meas. points in C (0...12)?:** 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

MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option) 19.4

- ▶ **Q423 Number of probes?:** 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.
- ▶ **Q431 Preset (0/1/2/3)?:** 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 before the cycle start via the calibration sphere
 - 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 before cycle start via the calibration sphere
- ▶ **Q432 Angular range of backlash comp.?:** 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.

Touch Probe Cycles: Automatic Kinematics Measurement

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

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.

"Optimize position of rotary axes" 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 and Angle Optimization mode Q406 = 2

- The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the TNC tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- Optimization of the angle is followed by optimization of the position. No additional measurements are necessary for this; the TNC calculates the position optimization 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)
 - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

Touch Probe Cycles: Automatic Kinematics Measurement

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

Cycle run

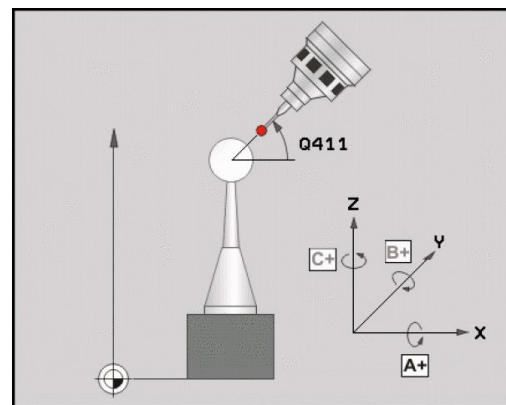
Touch Probe Cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)", page 644). Then the TNC corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration 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.



PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option) 19.5

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 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.



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.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option) 19.5

Cycle parameters



- ▶ **Q407 Radius of calib. sphere?:** Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- ▶ **Q320 Set-up clearance?** (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
- ▶ **Q408 Retraction height?** (absolute): Input range 0.0001 to 99999.9999
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
>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.
- ▶ **Q253 Feed rate for pre-positioning?:** Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively **FMAX, FAUTO, PREDEF**
- ▶ **Q380 Ref. angle in ref. axis?** (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
- ▶ **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q413 Angle of incidence in A axis?:** Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- ▶ **Q414 No. of meas. points in A (0...12)?:** 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

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

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- ▶ **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q417 Angle of incidence in B axis?:** Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- ▶ **Q418 No. of meas. points in B (0...12)?:** 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
- ▶ **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- ▶ **Q421 Angle of incidence in C axis?:** Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- ▶ **Q422 No. of meas. points in C (0...12)?:** 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
- ▶ **Q423 Number of probes?:** 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.
- ▶ **Q432 Angular range of backlash comp.?:** 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

PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option) 19.5

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

Touch Probe Cycles: Automatic Kinematics Measurement

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- ▶ 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

3 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**

PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option) 19.5

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

Touch Probe Cycles: Automatic Kinematics Measurement

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- ▶ 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

**5 TCH PROBE 452 PRESET
COMPENSATION**

Q407=12.5 ;SPHERE RADIUS

Q320=0 ;SET-UP CLEARANCE

Q408=0 ;RETR. HEIGHT

Q253=99999F 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
 - Position before preset-compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Position after preset-compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

Notes on log data

(see "Logging function", page 657)

20

**Touch Probe
Cycles: Automatic
Tool Measurement**

20.1 Fundamentals

20.1 Fundamentals

Overview



When running touch probe cycles, Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, 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:

New format	Old format	Cycle	Page
		Calibrating the TT, Cycles 30 and 480	676
		Calibrating the wireless TT 449, Cycle 484	677
		Measuring the tool length, Cycles 31 and 481	679
		Measuring the tool radius, Cycles 32 and 482	681
		Measuring the tool length and radius, Cycles 33 and 483	683



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**.

20.1 Fundamentals

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 the machine parameter **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:

$n = \text{maxPeriphSpeedMeas} / (r \cdot 0.0063)$ 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:

$v = \text{measuring tolerance} \cdot n$ 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 = $r \cdot \text{measureTolerance1} / 5 \text{ mm}$, where

r: Active tool radius in mm
measureTolerance1: Maximum permissible error of measurement

Touch Probe Cycles: Automatic Tool Measurement

20.1 Fundamentals

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)

Touch Probe Cycles: Automatic Tool Measurement

20.2 Calibrate the TT (Cycle 30 or 480, DIN/ISO: G480 Option 17)


20.2 Calibrate the TT (Cycle 30 or 480, DIN/ISO: G480 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 671). 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 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



- **Q260 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

Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484) 20.3

20.3 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484)

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. After the calibration, the TNC stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Touch Probe Cycles: Automatic Tool Measurement

20.3 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484)

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 parameters. 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



Q536 Stop before running (0=Stop)?: Specify whether to stop before cycle start or run the cycle automatically without stopping:
0: Stop before running the cycle. 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 cycle. 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

20.4 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481)

Cycle run

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 481 (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 contact is made. 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.

Touch Probe Cycles: Automatic Tool Measurement

20.4 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481)

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



- ▶ **Tool measurement mode (0-2)?:** Specify whether and how the determined data will be entered in the tool table
 - 0:** The measured tool length is written to column L of the tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value stored in TOOL.T, it will be overwritten.
 - 1:** The measured tool length is compared to the tool length L from TOOL.T. It then calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for 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)
 - 2:** The measured tool length is compared to the tool length L from TOOL.T. The TNC calculates the deviation from the stored value and enters it in Q parameter Q115. Nothing is entered under L or DL in the tool table.
- ▶ **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
- ▶ **Probe the teeth? 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 CAL. 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 CAL. 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 CAL. TOOL LENGTH
Q340=1 ;CHECK
Q260=+100;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH

20.5 Measuring tool radius (Cycle 32 or 482, DIN/ISO: G482)

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 671). 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.

Touch Probe Cycles: Automatic Tool Measurement

20.5 Measuring tool radius (Cycle 32 or 482, DIN/ISO: G482)

Cycle parameters



- ▶ **Tool measurement mode (0-2)?:** Specify whether and how the determined data will be entered in the tool table
0: The measured tool radius is written to column R of the tool table TOOL.T, and the tool compensation is set to DR=0. If there is already a value stored in TOOL.T, it will be overwritten.
1: The measured tool radius is compared to the tool radius R from TOOL.T. It then calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for 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)
2: The measured tool radius is compared to the tool radius R from TOOL.T. The TNC calculates the deviation from the stored value and enters it in Q parameter Q116. Nothing is entered under R or DR in the tool table.
- ▶ **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
- ▶ **Probe the teeth? 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 32.0 CAL. 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 CAL. 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 CAL. TOOL RADIUS
Q340=1 ;CHECK
Q260=+100;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH

20.6 Measuring tool length and radius (Cycle 33 or 483, DIN/ISO: G483)

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 671). 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.

Touch Probe Cycles: Automatic Tool Measurement

20.6 Measuring tool length and radius (Cycle 33 or 483, DIN/ISO: G483)

Cycle parameters



- ▶ **Tool measurement mode (0-2)?:** Specify whether and how the determined data will be entered in the tool table
0: The measured tool length and measured tool radius are written to columns L and R of the tool table TOOL.T, and the tool compensation is set to DL=0 and DR=0. If there is already a value stored in TOOL.T, it will be overwritten.
1: The measured tool length and measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The TNC calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in Q parameters Q115 and Q116. If the delta value is greater than the permissible tool length or radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T)
2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The TNC calculates the deviation from the stored value and enters them in Q parameters Q115 and Q116. Nothing is entered under L, R, DL, or DR in the tool table.
- ▶ **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
- ▶ **Probe the teeth? 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 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

21

Tables of Cycles

Tables of Cycles

21.1 Overview

21.1 Overview

Fixed cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
7	Zero point shift	■		269
8	Mirroring	■		276
9	Dwell time	■		293
10	Rotation	■		278
11	Scaling factor	■		280
12	Program call	■		294
13	Oriented spindle stop	■		296
14	Contour definition	■		204
19	Tilting the working plane	■		283
20	Contour data SL II	■		209
21	Pilot drilling SL II		■	211
22	Rough out SL II		■	213
23	Floor finishing SL II		■	217
24	Side finishing SL II		■	219
25	Contour train		■	222
26	Axis-specific scaling	■		281
27	Cylinder surface		■	237
28	Cylindrical surface slot		■	240
29	Cylinder surface ridge		■	243
32	Tolerance	■		297
39	Cylinder surface contour		■	246
200	Drilling		■	81
201	Reaming		■	83
202	Boring		■	85
203	Universal drilling		■	88
204	Back boring		■	91
205	Universal pecking		■	95
206	Tapping with a floating tap holder, new		■	111
207	Rigid tapping, new		■	114
208	Bore milling		■	99
209	Tapping with chip breaking		■	117
220	Polar pattern	■		193
221	Cartesian pattern	■		196
225	Engraving		■	318
232	Face milling		■	323

Cycle number	Cycle designation	DEF active	CALL active	Page
233	Face milling (selectable milling direction, consider the side walls)		■	180
239	Ascertain the load	■		328
240	Centering		■	79
241	Single-lip deep-hole drilling		■	102
247	Datum setting	■		275
251	Rectangular pocket (complete machining)		■	147
252	Circular pocket (complete machining)		■	152
253	Slot milling		■	157
254	Circular slot		■	162
256	Rectangular stud (complete machining)		■	167
257	Circular stud (complete machining)		■	171
258	Polygon stud		■	175
262	Thread milling		■	123
263	Thread milling/countersinking		■	127
264	Thread drilling/milling		■	131
265	Helical thread drilling/milling		■	135
267	Outside thread milling		■	139
270	Contour train data		■	224
275	Trochoidal slot		■	225
291	Coupling turning interpolation		■	300
292	Contour turning interpolation		■	311

Turning cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
800	Adapt rotary coordinate system	■		342
801	Reset the turning system	■		348
810	Turn contour, longitudinal		■	364
811	Turn shoulder, longitudinal		■	350
812	Turn shoulder, longitudinal extended		■	353
813	Turn, longitudinal plunge		■	357
814	Turn, longitudinal plunge extended		■	360
815	Turn contour-parallel		■	368
820	Turn contour, transverse		■	386
821	Turn shoulder face		■	372
822	Turn shoulder face extended		■	375
823	Turn, transverse plunge		■	379
824	Turn, transverse plunge extended		■	382
830	Thread, contour-parallel		■	445
831	Thread, longitudinal		■	438
832	Thread, extended		■	441
860	Recessing contour, radial		■	422
861	Recessing, radial		■	414
862	Recessing, radial extended		■	418
870	Recessing contour, axial		■	434
871	Recessing, axial		■	426
872	Recessing, axial extended		■	429

Touch probe cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
0	Reference plane	■		552
1	Polar datum	■		553
3	Measuring	■		593
4	Measuring in 3-D	■		595
444	Probing in 3D	■		597
30	Calibrate the TT	■		676
31	Measure/Inspect the tool length	■		679
32	Measure/Inspect the tool radius	■		681
33	Measure/Inspect the tool length and the tool radius	■		683
400	Basic rotation using two points	■		474
401	Basic rotation over two holes	■		477
402	Basic rotation over two studs	■		480
403	Compensate misalignment with rotary axis	■		483
404	Set basic rotation	■		486
405	Compensate misalignment with the C axis	■		487
408	Reference point at slot center (FCL 3 function)	■		496
409	Reference point at ridge center (FCL 3 function)	■		500
410	Datum from inside of rectangle	■		503
411	Datum from outside of rectangle	■		507
412	Datum from inside of circle (hole)	■		510
413	Datum from outside of circle (stud)	■		515
414	Datum from outside of corner	■		519
415	Datum from inside of corner	■		524
416	Datum from circle center	■		528
417	Datum in touch probe axis	■		532
418	Datum at center between four holes	■		534
419	Datum in any one axis	■		538
420	Workpiece—measure angle	■		554
421	Workpiece—measure hole (center and diameter of hole)	■		557
422	Workpiece—measure circle from outside (diameter of circular stud)	■		562
423	Workpiece—measure rectangle from inside	■		567
424	Workpiece—measure rectangle from outside	■		570
425	Workpiece—measure inside width (slot)	■		573
426	Workpiece—measure outside width (ridge)	■		576
427	Workpiece—measure in any selectable axis	■		579
430	Workpiece—measure bolt hole circle	■		582

Tables of Cycles

21.1 Overview

Cycle number	Cycle designation	DEF active	CALL active	Page
431	Workpiece—measure plane	■		582
450	KinematicsOpt: Save kinematics (option)	■		641
451	KinematicsOpt: Measure kinematics (option)	■		644
452	KinematicsOpt: Preset compensation	■		638
460	Calibrate the touch probe	■		604
461	Calibrate touch probe length	■		608
462	Calibrate touch probe inside radius	■		610
463	Calibrate touch probe outside radius	■		612
480	Calibrate the TT	■		676
481	Measure/Inspect the tool length	■		679
482	Measure/Inspect the tool radius	■		681
483	Measure/Inspect the tool length and the tool radius	■		683
484	Calibrate TT	■		677
600	Global workspace	■		626
601	Local workspace	■		631

Index

3

3D Touch Probes.....	462
3-D touch probes.....	54

A

Adapt rotary coordinate system.....	342
Automatic datum setting.....	492
At center of 4 holes.....	534
Center of a bolt hole circle.....	528
Center of a circular pocket (hole).....	510
Center of a circular stud.....	515
Center of a rectangular pocket.....	503
Center of a rectangular stud.....	507
In any axis.....	538
Inside of corner.....	524
In the touch probe axis.....	532
Outside of corner.....	519
Ridge center.....	500
Slot center.....	496
Automatic tool measurement.....	674
Axis-specific scaling.....	281

B

Back boring.....	91
Basic rotation	
Measure during program run.....	472
Blank form update.....	340
Bolt hole circle.....	193
Bore milling.....	99
Boring.....	85

C

Centering.....	79
Circular pocket	
Roughing+finishing.....	152
Circular point patterns.....	193
Circular slot	
Roughing+finishing.....	162
Circular stud.....	171
Classification of results.....	549
Compensating workpiece misalignment.....	472
By measuring two points on a straight line.....	474
Over two circular studs.....	480
Over two holes.....	477
Via rotary axis.....	483, 487
Consideration of basic rotation.....	462
Contour cycles.....	202
Contour train.....	222, 224
Coordinate transformation.....	268
COUPLING INTERPOLATION TURNING.....	311

Cycle.....	58
Calling.....	60
Define.....	59
Cycles and point tables.....	75
Cylinder surface	
Machine contour.....	237, 246
Ridge machining.....	243
Slot machining.....	240

D

Datum shift.....	269
In the program.....	269
With datum tables.....	270
Drilling.....	81, 88, 95
Drilling Cycles.....	78
Dwell time.....	293

E

Engraving.....	318
----------------	-----

F

Face milling.....	323
FCL function.....	9
Feature Content Level.....	9
Floor finishing.....	217
FUNCTION TURN DATA.....	340
Fundamentals of Thread Milling.....	121

G

GEAR HOBBING.....	449, 454
-------------------	----------

H

Helical thread drilling/milling.....	135
--------------------------------------	-----

I

Inside thread milling.....	123
INTERPOLATION TURNING, CONTOUR FINISHING.....	300

K

Kinematic measurement	
Accuracy.....	649
Backlash.....	651
Calibration methods.....	650, 663, 665
Hirth coupling.....	647
Logging function.....	642, 657, 667
Measuring point selection.....	643, 648
Measuring position selection.....	649
Prerequisites.....	640
Kinematics measurement.....	638
Kinematics Measurement	
Measure kinematics.....	644, 658
Kinematics measurement	
Save kinematics.....	641
KinematicsOpt.....	638

Linear point patterns.....	196
----------------------------	-----

M

Machine parameters for 3D touch probe.....	465
Machining pattern.....	66
Measure angle.....	554
Measure angle of a plane.....	585
Measure any coordinate.....	579
Measure bolt hole circle.....	582
Measure hole.....	557
Measure hole inside.....	557
Measure hole outside.....	562
Measure kinematics.....	644
Preset compensation.....	658
Measurement parameters.....	549
Measurement results in Q parameters.....	549
Measure rectangular pocket.....	570
Measure rectangular stud.....	567
Measure the plane angle.....	585
Measure the slot width.....	573
Measure the width of a ridge.....	576, 576
Measuring slot width.....	573
Measuring the width of a ridge.....	576
Mirroring.....	276

O

Outside thread milling.....	139
-----------------------------	-----

P

Pattern definition.....	66
Peck drilling.....	95, 102
Point patterns.....	192
Overview.....	192
Point tables.....	73
Polygon stud.....	175
Positioning logic.....	467
Probing feed rate.....	466
Program call.....	294
Via cycle.....	294

R

Reaming.....	83
Recording measurement results.....	547
Rectangular pocket	
Roughing+finishing.....	147
Rectangular stud.....	167
Reference images.....	616
Reset rotary coordinate system.....	348
Rotation.....	278
Roughing:See SL Cycles, Roughing.....	213

S

Scaling..... 280
 Set a basic rotation..... 486
 Side finishing..... 219
 Single-lip deep-hole drilling..... 102
 SL Cycles..... 202, 237, 246
 Contour cycle..... 204
 Contour data..... 209
 Contour train..... 222, 224
 Floor finishing..... 217
 SL cycles
 Fundamentals..... 202
 Fundamentals..... 264
 SL Cycles
 Pilot drilling..... 211
 Roughing..... 213
 Side finishing..... 219
 Superimposed contours. 205, 258
 SL cycles with complex contour
 formula..... 254
 SL cycles with simple contour
 formula..... 264
 Slot milling
 Roughing+finishing..... 157
 Spindle orientation..... 296

T

Tapping
 With a floating tap holder..... 111
 With chip breaking..... 117
 Without a floating tap
 holder..... 114, 117
 Thread drilling/milling..... 131
 Thread milling/countersinking... 127
 Tilting function
 Procedure..... 288
 Tilting the working plane.. 283, 283
 Cycle..... 283
 Tolerance monitoring..... 549
 Tool compensation..... 550
 Tool measurement..... 670, 674
 Calibrate TT..... 676, 677
 Machine parameters..... 672
 Measuring tool length and
 radius..... 683
 Tool length..... 679
 Tool radius..... 681
 Tool monitoring..... 550
 Touch probe cycles
 For automatic mode..... 464
 Touch probe data..... 469
 Touch probe table..... 468
 Turning cycles..... 336, 349
 Axial recessing....
 402, 410, 426, 434
 Contour face..... 386
 Contour longitudinal..... 364
 Contour-parallel..... 368

Radial recessing..... 390, 414
 Recessing, axial extended.... 405,
 429
 Recessing, radial extended....
 393, 418
 Recessing contour, radial.... 398,
 422
 shoulder, longitudinal extended....
 353
 Shoulder face..... 372
 Shoulder face extended..... 375
 Shoulder longitudinal..... 350
 Thread, contour-parallel..... 445
 Thread, extended..... 441
 Thread, longitudinal..... 438
 Transverse plunge..... 379
 Transverse plunge extended.. 382
 Turn, longitudinal plunge..... 357
 Turn, longitudinal plunge
 extended..... 360

U

Universal drilling..... 88, 95

W

Workpiece Measurement..... 546

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

E-mail: service.ms-support@heidenhain.de

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.

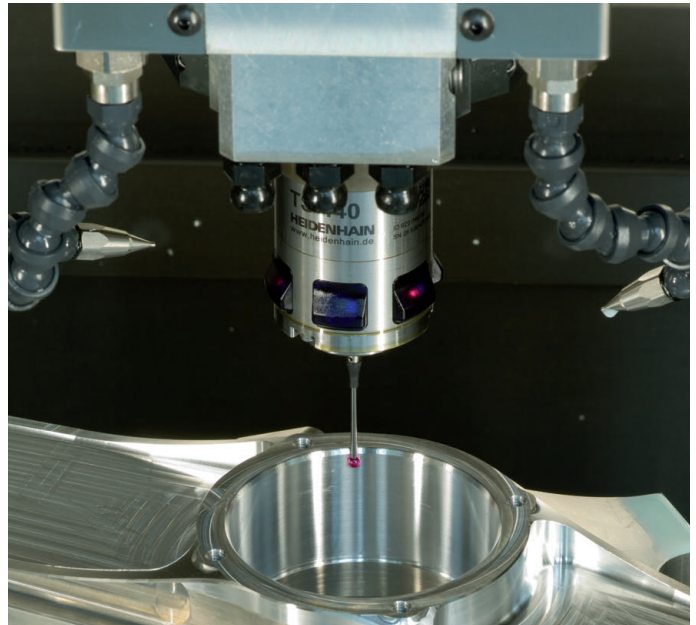
Workpiece touch probes

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

