About this Manual

The symbols used in this manual are described below.

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

This symbol indicates that there is one or more of the following risks when using the described function:
- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator

This symbol indicates 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 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.

<table>
<thead>
<tr>
<th>TNC model</th>
<th>NC software number</th>
</tr>
</thead>
<tbody>
<tr>
<td>iTNC 530</td>
<td>340 490-06</td>
</tr>
<tr>
<td>iTNC 530 E</td>
<td>340 491-06</td>
</tr>
<tr>
<td>iTNC 530</td>
<td>340 492-06</td>
</tr>
<tr>
<td>iTNC 530 E</td>
<td>340 493-06</td>
</tr>
<tr>
<td>iTNC 530 programming station</td>
<td>340 494-06</td>
</tr>
<tr>
<td>iTNC 530, HSCI and HeROS 5</td>
<td>606 420-01</td>
</tr>
<tr>
<td>iTNC 530, HSCI and HeROS 5</td>
<td>606 421-01</td>
</tr>
</tbody>
</table>

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

**HSCI** (HEIDENHAIN Serial Controller Interface) identifies the new hardware platform of the TNC controls.

**HeROS 5** identifies the new operating system of HSCI-based TNC controls.

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

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

- Tool measurement with the TT

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

**User's Manual:**

All TNC functions that have no connection with cycles are described in the User’s Manual of the iTNC 530. Please contact HEIDENHAIN if you require a copy of this User’s Manual.


**smarT.NC user documentation:**

The smarT.NC operating mode is described in a separate Pilot. Please contact HEIDENHAIN if you require a copy of this Pilot. ID: 533 191-xx.
**Software options**

The iTNC 530 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:

<table>
<thead>
<tr>
<th>Software option 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder surface interpolation (Cycles 27, 28, 29 and 39)</td>
</tr>
<tr>
<td>Feed rate in mm/min for rotary axes: <strong>M116</strong></td>
</tr>
<tr>
<td>Tilting the machining plane (Cycle 19, <strong>PLANE</strong> function and 3-D ROT soft key in the Manual operating mode)</td>
</tr>
<tr>
<td>Circle in 3 axes with tilted working plane</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Software option 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Block processing time 0.5 ms instead of 3.6 ms</td>
</tr>
<tr>
<td>5-axis interpolation</td>
</tr>
<tr>
<td>Spline interpolation</td>
</tr>
</tbody>
</table>

3-D machining:
- **M114**: Automatic compensation of machine geometry when working with swivel axes
- **M128**: Maintaining the position of the tool tip when positioning with tilted axes (TCPM)
- **FUNCTION TCPM**: Maintaining the position of the tool tip when positioning with tilted axes (TCPM) in selectable modes
- **M144**: Compensating the machine’s kinematic configuration for ACTUAL/NOMINAL positions at end of block
- Additional parameters for **finishing/roughing** and **tolerance for rotary axes** in Cycle 32 (G62)
- **LN** blocks (3-D compensation)

<table>
<thead>
<tr>
<th>DCM Collision software option</th>
</tr>
</thead>
<tbody>
<tr>
<td>Function which dynamically monitors areas defined by the machine manufacturer to prevent collisions.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Additional Conversational Language software option</th>
</tr>
</thead>
<tbody>
<tr>
<td>Function for enabling the conversational languages Slovenian, Slovak, Norwegian, Latvian, Estonian, Korean, Turkish, Romanian, Lithuanian.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DXF Converter software option</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extract contours from DXF files (R12 format).</td>
</tr>
<tr>
<td>Software Option</td>
</tr>
<tr>
<td>-----------------------------------</td>
</tr>
<tr>
<td><strong>Global Program Settings</strong></td>
</tr>
<tr>
<td><strong>AFC</strong></td>
</tr>
<tr>
<td><strong>KinematicsOpt</strong></td>
</tr>
<tr>
<td><strong>3D-ToolComp</strong></td>
</tr>
</tbody>
</table>
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.

<table>
<thead>
<tr>
<th>FCL 4 functions</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Graphical depiction of the protected space when DCM collision monitoring is active</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>Handwheel superimposition in stopped condition when DCM collision monitoring is active</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>3-D basic rotation (set-up compensation)</td>
<td>Machine manual</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>FCL 3 functions</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Touch probe cycle for 3-D probing</td>
<td>Page 451</td>
</tr>
<tr>
<td>Touch probe cycles for automatic datum setting using the center of a slot/ridge</td>
<td>Page 345</td>
</tr>
<tr>
<td>Feed-rate reduction for the machining of contour pockets with the tool being in full contact with the workpiece</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>PLANE function: Entry of axis angle</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>User documentation as a context-sensitive help system</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>smarT.NC: Programming of smarT.NC and machining can be carried out simultaneously</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>smarT.NC: Contour pocket on point pattern</td>
<td>smarT.NC Pilot</td>
</tr>
</tbody>
</table>
## 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.

<table>
<thead>
<tr>
<th><strong>FCL 3 functions</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>smarT.NC: Preview of contour programs in the file manager</td>
<td>smarT.NC Pilot</td>
</tr>
<tr>
<td>smarT.NC: Positioning strategy for machining point patterns</td>
<td>smarT.NC Pilot</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>FCL 2 functions</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>3-D line graphics</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>Virtual tool axis</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>Filtering of externally created contours</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>Possibility of assigning different depths to each subcontour in the contour formula</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>DHCP dynamic IP-address management</td>
<td>User’s Manual</td>
</tr>
<tr>
<td>Touch-probe cycle for global setting of touch-probe parameters</td>
<td>Page 456</td>
</tr>
<tr>
<td>smarT.NC: Graphic support of block scan</td>
<td>smarT.NC Pilot</td>
</tr>
<tr>
<td>smarT.NC: Coordinate transformation</td>
<td>smarT.NC Pilot</td>
</tr>
<tr>
<td>smarT.NC: PLANE function</td>
<td>smarT.NC Pilot</td>
</tr>
</tbody>
</table>
New Cycle Functions of Software
340 49x-02

- New machine parameter for defining the positioning speed (see “Touch trigger probe, rapid traverse for positioning: MP6151” on page 317)
- New machine parameter for consideration of basic rotation in Manual Operation (see “Consider a basic rotation in the Manual Operation mode: MP6166” on page 316)
- Cycles 420 to 431 for automatic tool measurement were improved so that the measuring log can now also be displayed on the screen (see “Recording the results of measurement” on page 397)
- A new cycle that enables you to set global touch probe parameters was introduced (see “FAST PROBING (Cycle 441, DIN/ISO: G441, FCL 2 Function)” on page 456)
New Cycle Functions of Software 340 49x-03

- New cycle for setting a datum in the center of a slot (see “SLOT CENTER REF PT (Cycle 408, DIN/ISO: G408, FCL 3 Function)” on page 345)
- New cycle for setting a datum in the center of a ridge (see “RIDGE CENTER REF PT (Cycle 409, DIN/ISO: G409, FCL 3 function)” on page 349)
- New 3-D probing cycle (see “MEASURING IN 3-D (Cycle 4, FCL 3 function)” on page 451)
- Cycle 401 now also allows you to compensate workpiece misalignment by rotating the rotary table (see “BASIC ROTATION from Two Holes (Cycle 401, DIN/ISO: G401)” on page 325)
- Cycle 402 now also allows you to compensate workpiece misalignment by rotating the rotary table (see “BASIC ROTATION over Two Studs (Cycle 402, DIN/ISO: G402)” on page 328)
- In the cycles for datum setting, the results of measurement are available in the Q parameters $Q_{15X}$ (see “Measurement results in Q parameters” on page 399)
New Cycle Functions of Software 340 49x-04

- New cycle for saving a machine's kinematic configuration (see “SAVE KINEMATICS (Cycle 450, DIN/ISO: G450; Option)” on page 464)
- New cycle for testing and optimizing a machine's kinematic configuration (see “MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451; Option)” on page 466)
- Cycle 412: Number of measuring points selectable through parameter Q423 (see “DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)” on page 360)
- Cycle 413: Number of measuring points selectable through parameter Q423 (see “DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)” on page 364)
- Cycle 421: Number of measuring points selectable through parameter Q423 (see “MEASURE HOLE (Cycle 421, DIN/ISO: G421)” on page 408)
- Cycle 422: Number of measuring points selectable through parameter Q423 (see “MEAS. CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422)” on page 412)
- Cycle 3: Error message can be suppressed if the stylus is already deflected when a cycle starts (see “MEASURING (Cycle 3)” on page 449)
- New cycle for rectangular stud milling (see “RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)” on page 160)
- New cycle for circular stud milling (see “CIRCULAR STUD (Cycle 257, DIN/ISO: G257)” on page 164)
New Cycle Functions of Software
340 49x-05

- New machining cycle for single-fluted deep-hole drilling (see “SINGLE-FLUTED DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)” on page 98)
- Touch Probe Cycle 404 (SET BASIC ROTATION) was expanded by parameter Q305 (Number in table) in order to write basic rotations to the preset table (see page 335)
- Touch Probe Cycles 408 to 419: The TNC now also writes to line 0 of the preset table when the display value is set (see “Saving the calculated datum” on page 344)
- Touch Probe Cycle 412: Additional parameter Q365 “type of traverse” (see “DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)” on page 360)
- Touch Probe Cycle 413: Additional parameter Q365 “type of traverse” (see “DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)” on page 364)
- Touch Probe Cycle 416: Additional parameter Q320 (setup clearance, see “DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)”, page 377)
- Touch Probe Cycle 421: Additional parameter Q365 “type of traverse” (see “MEASURE HOLE (Cycle 421, DIN/ISO: G421)” on page 408)
- Touch Probe Cycle 422: Additional parameter Q365 “type of traverse” (see “MEAS. CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422)” on page 412)
- Touch Probe Cycle 425 (MEASURE SLOT) was expanded by parameters Q301 (Move to clearance height) and Q320 (setup clearance) (see “MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)”, page 424)
- Touch Probe Cycle 450 (SAVE KINEMATICS) was expanded by input option 2 (Display saving status) in parameter Q410 (mode) (see “SAVE KINEMATICS (Cycle 450, DIN/ISO: G450; Option)” on page 464)
- Touch Probe Cycle 451 (MEASURE KINEMATICS) was expanded by parameters Q423 (Number of circular measurements) and Q432 (Set preset) (see “Cycle parameters” on page 475)
- New Touch Probe Cycle 452 (PRESET COMPENSATION) simplifies the measurement of tool changer heads (see “PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, Option)” on page 482)
- New Touch Probe Cycle 484 for calibrating the wireless TT 449 tool touch probe (see “CALIBRATING THE WIRELESS TT 449 (Cycle 484, DIN/ISO: G484)” on page 500)
New Cycle Functions of Software 340 49x-06

- New Cycle 275 “Trochoidal Contour Slot” (see “TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)” on page 205)
- In Cycle 241 “Single-Fluted Deep-Hole Drilling” it is now possible to define a dwell depth (see “SINGLE-FLUTED DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)” on page 98)
- The approach and departure behavior of Cycle 39 “Cylinder Surface Contour” can now be adjusted (see “Cycle run” on page 230)
- New touch probe cycle for calibration of a touch probe on a calibration sphere (see “CALIBRATE TS (Cycle 460, DIN/ISO: G460)” on page 458)
- KinematicsOpt: An additional parameter for determination of the backlash in a rotary axis was introduced (see “Backlash” on page 473)
- KinematicsOpt: Better support for positioning of Hirth-coupled axes (see “Machines with Hirth-coupled axes” on page 469)
Cycle Functions Changed Since the Predecessor Versions
340 422-xx/340 423-xx

- The management of more than one block of calibration data was changed (see Conversational Programming User’s Manual).
The cylindrical surface cycles 27, 28, 29 and 39 can now also be used with modulo rotary axes. In the past, Machine Parameter 810.x = 0 was required.

Cycle 403 does not check whether touch points and compensation axis match. As a result, probing is also possible in a tilted coordinate system (see “BASIC ROTATION Compensation via Rotary Axis (Cycle 403, DIN/ISO: G403)” on page 331)
The approach behavior during side finishing with Cycle 24 (DIN/ISO: G124) was changed (see “Please note while programming:” on page 199)
<table>
<thead>
<tr>
<th>Contents</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fundamentals / Overviews</td>
<td>1</td>
</tr>
<tr>
<td>Using Cycles</td>
<td>2</td>
</tr>
<tr>
<td>Fixed Cycles: Drilling</td>
<td>3</td>
</tr>
<tr>
<td>Fixed Cycles: Tapping / Thread Milling</td>
<td>4</td>
</tr>
<tr>
<td>Fixed Cycles: Pocket Milling / Stud Milling</td>
<td>5</td>
</tr>
<tr>
<td>Fixed Cycles: Pattern Definitions</td>
<td>6</td>
</tr>
<tr>
<td>Fixed Cycles: Contour Pocket</td>
<td>7</td>
</tr>
<tr>
<td>Fixed Cycles: Cylindrical Surface</td>
<td>8</td>
</tr>
<tr>
<td>Fixed Cycles: Contour Pocket with Contour Formula</td>
<td>9</td>
</tr>
<tr>
<td>Fixed Cycles: Multipass Milling</td>
<td>10</td>
</tr>
<tr>
<td>Cycles: Coordinate Transformations</td>
<td>11</td>
</tr>
<tr>
<td>Cycles: Special Functions</td>
<td>12</td>
</tr>
<tr>
<td>Using Touch Probe Cycles</td>
<td>13</td>
</tr>
<tr>
<td>Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment</td>
<td>14</td>
</tr>
<tr>
<td>Touch Probe Cycles: Automatic Datum Setting</td>
<td>15</td>
</tr>
<tr>
<td>Touch Probe Cycles: Automatic Workpiece Inspection</td>
<td>16</td>
</tr>
<tr>
<td>Touch Probe Cycles: Special Functions</td>
<td>17</td>
</tr>
<tr>
<td>Touch Probe Cycles: Automatic Kinematics Measurement</td>
<td>18</td>
</tr>
<tr>
<td>Touch Probe Cycles: Automatic Tool Measurement</td>
<td>19</td>
</tr>
</tbody>
</table>
1 Fundamentals / Overviews ..... 43

1.1 Introduction ..... 44
1.2 Available Cycle Groups ..... 45
  Overview of fixed cycles ..... 45
  Overview of touch probe cycles ..... 46
2 Using Fixed Cycles ..... 47

2.1 Working with Fixed Cycles ..... 48
   Machine-specific cycles ..... 48
   Defining a cycle using soft keys ..... 49
   Defining a cycle using the GOTO function ..... 49
   Calling cycles ..... 50
   Working with the secondary axes U/V/W ..... 52

2.2 Program Defaults for Cycles ..... 53
   Overview ..... 53
   Entering GLOBAL DEF ..... 54
   Using GLOBAL DEF information ..... 54
   Global data valid everywhere ..... 55
   Global data for drilling operations ..... 55
   Global data for milling operations with pocket cycles 25x ..... 56
   Global data for milling operations with contour cycles ..... 56
   Global data for positioning behavior ..... 56
   Global data for probing functions ..... 57

2.3 Pattern Definition PATTERN DEF ..... 58
   Application ..... 58
   Entering PATTERN DEF ..... 59
   Using PATTERN DEF ..... 59
   Defining individual machining positions ..... 60
   Defining a single row ..... 61
   Defining a single pattern ..... 62
   Defining individual frames ..... 63
   Defining a full circle ..... 64
   Defining a circular arc ..... 65

2.4 Point Tables ..... 66
   Function ..... 66
   Creating a point table ..... 66
   Hiding single points from the machining process ..... 67
   Selecting a point table in the program ..... 68
   Calling a cycle in connection with point tables ..... 69
3 Fixed Cycles: Drilling ..... 71

3.1 Basics ..... 72
  Overview ..... 72
3.2 CENTERING (Cycle 240, DIN/ISO: G240) ..... 73
  Cycle run ..... 73
  Please note while programming: ..... 73
  Cycle parameters ..... 74
3.3 DRILLING (Cycle 200) ..... 75
  Cycle run ..... 75
  Please note while programming: ..... 75
  Cycle parameters ..... 76
3.4 REAMING (Cycle 201, DIN/ISO: G201) ..... 77
  Cycle run ..... 77
  Please note while programming: ..... 77
  Cycle parameters ..... 78
3.5 BORING (Cycle 202, DIN/ISO: G202) ..... 79
  Cycle run ..... 79
  Please note while programming: ..... 80
  Cycle parameters ..... 81
3.6 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203) ..... 83
  Cycle run ..... 83
  Please note while programming: ..... 84
  Cycle parameters ..... 85
3.7 BACK BORING (Cycle 204, DIN/ISO: G204) ..... 87
  Cycle run ..... 87
  Please note while programming: ..... 88
  Cycle parameters ..... 89
3.8 UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205) ..... 91
  Cycle run ..... 91
  Please note while programming: ..... 92
  Cycle parameters ..... 93
3.9 BORE MILLING (Cycle 208) ..... 95
  Cycle run ..... 95
  Please note while programming: ..... 96
  Cycle parameters ..... 97
3.10 SINGLE-FLUTED DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241) ..... 98
  Cycle run ..... 98
  Please note while programming: ..... 98
  Cycle parameters ..... 99
3.11 Programming Examples ..... 101
4 Fixed Cycles: Tapping / Thread Milling ..... 105

4.1 Basics ..... 106
  Overview ..... 106

4.2 TAPPING NEW with a Floating Tap Holder (Cycle 206, DIN/ISO: G206) ..... 107
  Cycle run ..... 107
  Please note while programming: ..... 107
  Cycle parameters ..... 108

4.3 RIGID TAPPING without a Floating Tap Holder NEW (Cycle 207, DIN/ISO: G207) ..... 109
  Cycle run ..... 109
  Please note while programming: ..... 110
  Cycle parameters ..... 111

4.4 TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209) ..... 112
  Cycle run ..... 112
  Please note while programming: ..... 113
  Cycle parameters ..... 114

4.5 Fundamentals of Thread Milling ..... 115
  Prerequisites ..... 115

4.6 THREAD MILLING (Cycle 262, DIN/ISO: G262) ..... 117
  Cycle run ..... 117
  Please note while programming: ..... 118
  Cycle parameters ..... 119

4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263) ..... 120
  Cycle run ..... 120
  Please note while programming: ..... 121
  Cycle parameters ..... 122

4.8 THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264) ..... 124
  Cycle run ..... 124
  Please note while programming: ..... 125
  Cycle parameters ..... 126

4.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265) ..... 128
  Cycle run ..... 128
  Please note while programming: ..... 129
  Cycle parameters ..... 130

4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267) ..... 132
  Cycle run ..... 132
  Please note while programming: ..... 133
  Cycle parameters ..... 134

4.11 Programming Examples ..... 136
5 Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling ..... 139

5.1 Basics ..... 140
   Overview ..... 140

5.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251) ..... 141
   Cycle run ..... 141
   Please note while programming: ..... 142
   Cycle parameters ..... 143

5.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252) ..... 146
   Cycle run ..... 146
   Please note while programming: ..... 147
   Cycle parameters ..... 148

5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253) ..... 150
   Cycle run ..... 150
   Please note while programming: ..... 151
   Cycle parameters ..... 152

5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254) ..... 155
   Cycle run ..... 155
   Please note while programming: ..... 156
   Cycle parameters ..... 157

5.6 RECTANGULAR STUD (Cycle 256, DIN/ISO: G256) ..... 160
   Cycle run ..... 160
   Please note while programming: ..... 161
   Cycle parameters ..... 162

5.7 CIRCULAR STUD (Cycle 257, DIN/ISO: G257) ..... 164
   Cycle run ..... 164
   Please note while programming: ..... 165
   Cycle parameters ..... 166

5.8 Programming Examples ..... 168
6 Fixed Cycles: Pattern Definitions ..... 171

6.1 Fundamentals ..... 172
   Overview ..... 172

6.2 CIRCULAR PATTERN (Cycle 220, DIN/ISO: G220) ..... 173
   Cycle run ..... 173
   Please note while programming: ..... 173
   Cycle parameters ..... 174

6.3 LINEAR PATTERN (Cycle 221, DIN/ISO: G221) ..... 176
   Cycle run ..... 176
   Please note while programming: ..... 176
   Cycle parameters ..... 177

6.4 Programming Examples ..... 178
7 Fixed Cycles: Contour Pocket, Contour Trains ..... 181

7.1 SL Cycles ..... 182
  Fundamentals ..... 182
  Overview ..... 184

7.2 CONTOUR GEOMETRY (Cycle 14, DIN/ISO: G37) ..... 185
  Please note while programming: ..... 185
  Cycle parameters ..... 185

7.3 Overlapping Contours ..... 186
  Basics ..... 186
  Subprograms: overlapping pockets ..... 187
  Area of inclusion ..... 188
  Area of exclusion ..... 189
  Area of intersection ..... 189

7.4 CONTOUR DATA (Cycle 20, DIN/ISO: G120) ..... 190
  Please note while programming: ..... 190
  Cycle parameters ..... 191

7.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121) ..... 192
  Cycle run ..... 192
  Please note while programming: ..... 192
  Cycle parameters ..... 193

7.6 ROUGH-OUT (Cycle 22, DIN/ISO: G122) ..... 194
  Cycle run ..... 194
  Please note while programming: ..... 195
  Cycle parameters ..... 196

7.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123) ..... 198
  Cycle run ..... 198
  Please note while programming: ..... 198
  Cycle parameters ..... 198

7.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124) ..... 199
  Cycle run ..... 199
  Please note while programming: ..... 199
  Cycle parameters ..... 200

7.9 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125) ..... 201
  Cycle run ..... 201
  Please note while programming: ..... 201
  Cycle parameters ..... 202

7.10 CONTOUR DATA (Cycle 270, DIN/ISO: G270) ..... 203
  Please note while programming: ..... 203
  Cycle parameters ..... 204

7.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275) ..... 205
  Cycle run ..... 205
  Please note while programming: ..... 207
  Cycle parameters ..... 208

7.12 Programming Examples ..... 211
8 Fixed Cycles: Cylindrical Surface ..... 219

8.1 Basics ..... 220
  Overview of cylindrical surface cycles ..... 220
8.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, Software Option 1) ..... 221
  Execution of cycle ..... 221
  Please note while programming: ..... 222
  Cycle parameters ..... 223
8.3 CYLINDER SURFACE Slot Milling (Cycle 28, DIN/ISO: G128, Software Option 1) ..... 224
  Cycle run ..... 224
  Please note while programming: ..... 225
  Cycle parameters ..... 226
8.4 CYLINDER SURFACE Ridge Milling (Cycle 29, DIN/ISO: G129, Software Option 1) ..... 227
  Cycle run ..... 227
  Please note while programming: ..... 228
  Cycle parameters ..... 229
8.5 CYLINDER SURFACE Outside Contour Milling (Cycle 39, DIN/ISO: G139, Software Option 1) ..... 230
  Cycle run ..... 230
  Please note while programming: ..... 231
  Cycle parameters ..... 232
8.6 Programming Examples ..... 233
9 Fixed Cycles: Contour Pocket with Contour Formula ..... 237

9.1 SL Cycles with Complex Contour Formula ..... 238
   Basics ..... 238
   Selecting a program with contour definitions ..... 240
   Defining contour descriptions ..... 241
   Entering a complex contour formula ..... 242
   Overlapping contours ..... 243
   Contour machining with SL Cycles ..... 245

9.2 SL Cycles with Simple Contour Formula ..... 249
   Fundamentals ..... 249
   Entering a simple contour formula ..... 251
   Contour machining with SL Cycles ..... 251
10 Fixed Cycles: Multipass Milling ..... 253

10.1 Basics ..... 254
   Overview ..... 254
10.2 RUN 3-D DATA (Cycle 30, DIN/ISO: G60) ..... 255
   Cycle run ..... 255
   Please note while programming: ..... 255
   Cycle parameters ..... 256
10.3 MULTIPASS MILLING (Cycle 230, DIN/ISO: G230) ..... 257
   Cycle run ..... 257
   Please note while programming: ..... 257
   Cycle parameters ..... 258
10.4 RULED SURFACE (Cycle 231, DIN/ISO: G231) ..... 259
   Cycle run ..... 259
   Please note while programming: ..... 260
   Cycle parameters ..... 261
10.5 FACE MILLING (Cycle 232, DIN/ISO: G232) ..... 263
   Cycle run ..... 263
   Please note while programming: ..... 264
   Cycle parameters ..... 265
10.6 Programming Examples ..... 268
11 Cycles: Coordinate Transformations ..... 271

11.1 Fundamentals ..... 272
   Overview ..... 272
   Effect of coordinate transformations ..... 273
11.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54) ..... 274
   Effect ..... 274
   Cycle parameters ..... 274
11.3 DATUM Shift with Datum Tables (Cycle 7, DIN/ISO: G53) ..... 275
   Effect ..... 275
   Please note while programming: ..... 276
   Cycle parameters ..... 277
   Selecting a datum table in the part program ..... 277
   Editing the datum table in the Programming and Editing mode of operation ..... 278
   Editing a pocket table in a Program Run operating mode ..... 279
   Transferring the actual values into the datum table ..... 279
   Configuring the datum table ..... 280
   To exit a datum table ..... 280
11.4 DATUM SETTING (Cycle 247, DIN/ISO: G247) ..... 281
   Effect ..... 281
   Please note before programming: ..... 281
   Cycle parameters ..... 281
11.5 MIRROR IMAGE (Cycle 8, DIN/ISO: G28) ..... 282
   Effect ..... 282
   Please note while programming: ..... 282
   Cycle parameters ..... 283
11.6 ROTATION (Cycle 10, DIN/ISO: G73) ..... 284
   Effect ..... 284
   Please note while programming: ..... 284
   Cycle parameters ..... 285
11.7 SCALING (Cycle 11, DIN/ISO: G72) ..... 286
   Effect ..... 286
   Cycle parameters ..... 287
11.8 AXIS-SPECIFIC SCALING (Cycle 26) ..... 288
   Effect ..... 288
   Please note while programming: ..... 288
   Cycle parameters ..... 289
11.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, Software Option 1) ..... 290
   Effect ..... 290
   Please note while programming: ..... 291
   Cycle parameters ..... 292
   Resetting ..... 292
   Positioning the axes of rotation ..... 293
   Position display in the tilted system ..... 295
   Workspace monitoring ..... 295
   Positioning in a tilted coordinate system ..... 295
   Combining coordinate transformation cycles ..... 296
   Automatic workpiece measurement in the tilted system ..... 296
   Procedure for working with Cycle 19 WORKING PLANE ..... 297

11.10 Programming Examples ..... 299
12 Cycles: Special Functions ..... 301

12.1 Fundamentals ..... 302
  Overview ..... 302

12.2 DWELL TIME (Cycle 9, DIN/ISO: G04) ..... 303
  Function ..... 303
  Cycle parameters ..... 303

12.3 PROGRAM CALL (Cycle 12, DIN/ISO: G39) ..... 304
  Cycle function ..... 304
  Please note while programming: ..... 304
  Cycle parameters ..... 305

12.4 ORIENTED SPINDLE STOP (Cycle 13, DIN/ISO: G36) ..... 306
  Cycle function ..... 306
  Please note while programming: ..... 306
  Cycle parameters ..... 306

12.5 TOLERANCE (Cycle 32, DIN/ISO: G62) ..... 307
  Cycle function ..... 307
  Influences of the geometry definition in the CAM system ..... 308
  Please note while programming: ..... 309
  Cycle parameters ..... 310
13 Using Touch Probe Cycles ..... 311

13.1 General Information about Touch Probe Cycles ..... 312
  Method of function ..... 312
  Cycles in the Manual and El. Handwheel modes ..... 313
  Touch probe cycles for automatic operation ..... 313

13.2 Before You Start Working with Touch Probe Cycles ..... 315
  Maximum traverse to touch point: MP6130 ..... 315
  Safety clearance to touch point: MP6140 ..... 315
  Orient the infrared touch probe to the programmed probe direction: MP6165 ..... 315
  Consider a basic rotation in the Manual Operation mode: MP6166 ..... 316
  Multiple measurement: MP6170 ..... 316
  Confidence interval for multiple measurement: MP6171 ..... 316
  Touch trigger probe, probing feed rate: MP6120 ..... 317
  Touch trigger probe, rapid traverse for positioning: MP6150 ..... 317
  Touch trigger probe, rapid traverse for positioning: MP6151 ..... 317
  KinematicsOpt: Tolerance limit in Optimization mode: MP6600 ..... 317
  KinematicsOpt, permissible deviation of the calibration ball radius: MP6601 ..... 317
  Executing touch probe cycles ..... 318
14 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment ..... 319

14.1 Fundamentals ..... 320
   Overview ..... 320
   Characteristics common to all touch probe cycles for measuring workpiece misalignment ..... 321
14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400) ..... 322
   Cycle run ..... 322
   Please note while programming: ..... 322
   Cycle parameters ..... 323
14.3 BASIC ROTATION from Two Holes (Cycle 401, DIN/ISO: G401) ..... 325
   Cycle run ..... 325
   Please note while programming: ..... 325
   Cycle parameters ..... 326
14.4 BASIC ROTATION over Two Studs (Cycle 402, DIN/ISO: G402) ..... 328
   Cycle run ..... 328
   Please note while programming: ..... 328
   Cycle parameters ..... 329
14.5 BASIC ROTATION Compensation via Rotary Axis (Cycle 403, DIN/ISO: G403) ..... 331
   Cycle run ..... 331
   Please note while programming: ..... 332
   Cycle parameters ..... 333
14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404) ..... 335
   Cycle run ..... 335
   Cycle parameters ..... 335
14.7 Compensating Workpiece Misalignment by Rotating the C Axis (Cycle 405, DIN/ISO: G405) ..... 336
   Cycle run ..... 336
   Please note while programming: ..... 337
   Cycle parameters ..... 338
15 Touch Probe Cycles: Automatic Datum Setting ..... 341

15.1 Fundamentals ..... 342
   Overview ..... 342
   Characteristics common to all touch probe cycles for datum setting ..... 343
15.2 SLOT CENTER REF PT (Cycle 408, DIN/ISO: G408, FCL 3 Function) ..... 345
   Cycle run ..... 345
   Please note while programming: ..... 346
   Cycle parameters ..... 346
15.3 RIDGE CENTER REF PT (Cycle 409, DIN/ISO: G409, FCL 3 function) ..... 349
   Cycle run ..... 349
   Please note while programming: ..... 349
   Cycle parameters ..... 350
15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410) ..... 352
   Cycle run ..... 352
   Please note while programming: ..... 353
   Cycle parameters ..... 353
15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411) ..... 356
   Cycle run ..... 356
   Please note while programming: ..... 357
   Cycle parameters ..... 357
15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412) ..... 360
   Cycle run ..... 360
   Please note while programming: ..... 361
   Cycle parameters ..... 361
15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413) ..... 364
   Cycle run ..... 364
   Please note while programming: ..... 365
   Cycle parameters ..... 365
15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414) ..... 368
   Cycle run ..... 368
   Please note while programming: ..... 369
   Cycle parameters ..... 370
15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415) ..... 373
   Cycle run ..... 373
   Please note while programming: ..... 374
   Cycle parameters ..... 374
15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416) ..... 377
  Cycle run ..... 377
  Please note while programming: ..... 378
  Cycle parameters ..... 378
15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417) ..... 381
  Cycle run ..... 381
  Please note while programming: ..... 381
  Cycle parameters ..... 382
15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418) ..... 383
  Cycle run ..... 383
  Please note while programming: ..... 384
  Cycle parameters ..... 384
15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419) ..... 387
  Cycle run ..... 387
  Please note while programming: ..... 387
  Cycle parameters ..... 388
16 Touch Probe Cycles: Automatic Workpiece Inspection ..... 395

16.1 Fundamentals ..... 396
  Overview ..... 396
  Recording the results of measurement ..... 397
  Measurement results in Q parameters ..... 399
  Classification of results ..... 399
  Tolerance monitoring ..... 400
  Tool monitoring ..... 400
  Reference system for measurement results ..... 401

16.2 REF. PLANE (Cycle 0, DIN/ISO: G55) ..... 402
  Cycle run ..... 402
  Please note while programming: ..... 402
  Cycle parameters ..... 402

16.3 POLAR REFERENCE PLANE (Cycle 1) ..... 403
  Cycle run ..... 403
  Please note while programming: ..... 403
  Cycle parameters ..... 404

16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420) ..... 405
  Cycle run ..... 405
  Please note while programming: ..... 405
  Cycle parameters ..... 406

16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421) ..... 408
  Cycle run ..... 408
  Please note while programming: ..... 408
  Cycle parameters ..... 409

16.6 MEAS. CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422) ..... 412
  Cycle run ..... 412
  Please note while programming: ..... 412
  Cycle parameters ..... 413

16.7 MEAS. RECTAN. INSIDE (Cycle 423, DIN/ISO: G423) ..... 416
  Cycle run ..... 416
  Please note while programming: ..... 417
  Cycle parameters ..... 417

16.8 MEAS. RECTAN. OUTSIDE (Cycle 424, DIN/ISO: G424) ..... 420
  Cycle run ..... 420
  Please note while programming: ..... 421
  Cycle parameters ..... 421

16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425) ..... 424
  Cycle run ..... 424
  Please note while programming: ..... 424
  Cycle parameters ..... 425
16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426) ..... 427
   Cycle run ..... 427
   Please note while programming: ..... 427
   Cycle parameters ..... 428

16.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427) ..... 430
   Cycle run ..... 430
   Please note while programming: ..... 430
   Cycle parameters ..... 431

16.12 MEAS. BOLT HOLE CIRC. (Cycle 430, DIN/ISO: G430) ..... 433
   Cycle run ..... 433
   Please note while programming: ..... 433
   Cycle parameters ..... 434

16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431) ..... 437
   Cycle run ..... 437
   Please note while programming: ..... 438
   Cycle parameters ..... 439

16.14 Programming Examples ..... 441
17 Touch Probe Cycles: Special Functions ..... 445

17.1 Basics ..... 446
   Overview ..... 446
17.2 CALIBRATE TS (Cycle 2) ..... 447
   Cycle run ..... 447
   Please note while programming: ..... 447
   Cycle parameters ..... 447
17.3 CALIBRATE TS LENGTH (Cycle 9) ..... 448
   Cycle run ..... 448
   Cycle parameters ..... 448
17.4 MEASURING (Cycle 3) ..... 449
   Cycle run ..... 449
   Please note while programming: ..... 449
   Cycle parameters ..... 450
17.5 MEASURING IN 3-D (Cycle 4, FCL 3 function) ..... 451
   Cycle run ..... 451
   Please note while programming: ..... 451
   Cycle parameters ..... 452
17.6 MEASURE AXIS SHIFT (Touch Probe Cycle 440, DIN/ISO: G440) ..... 453
   Cycle run ..... 453
   Please note while programming: ..... 454
   Cycle parameters ..... 455
17.7 FAST PROBING (Cycle 441, DIN/ISO: G441, FCL 2 Function) ..... 456
   Cycle run ..... 456
   Please note while programming: ..... 456
   Cycle parameters ..... 457
17.8 CALIBRATE TS (Cycle 460, DIN/ISO: G460) ..... 458
   Cycle run ..... 458
   Please note while programming: ..... 458
   Cycle parameters ..... 459
18 Touch Probe Cycles: Automatic Kinematics Measurement ..... 461

18.1 Kinematic Measurement with TS Touch Probes (Option KinematicsOpt) ..... 462
   Fundamentals ..... 462
   Overview ..... 462
18.2 Prerequisites ..... 463
   Please note while programming: ..... 463
18.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450; Option) ..... 464
   Cycle run ..... 464
   Please note while programming: ..... 464
   Cycle parameters ..... 465
   Log function ..... 465
18.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451; Option) ..... 466
   Cycle run ..... 466
   Positioning direction ..... 468
   Machines with Hirth-coupled axes ..... 469
   Choice of number of measuring points ..... 470
   Choice of the calibration sphere position on the machine table ..... 470
   Notes on the accuracy ..... 471
   Notes on various calibration methods ..... 472
   Backlash ..... 473
   Please note while programming: ..... 474
   Cycle parameters ..... 475
   Various modes (Q406) ..... 478
   Log function ..... 479
18.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, Option) ..... 482
   Cycle run ..... 482
   Please note while programming: ..... 484
   Cycle parameters ..... 485
   Adjustment of tool changer heads ..... 487
   Drift Compensation ..... 489
   Log function ..... 491
19 Touch Probe Cycles: Automatic Tool Measurement ..... 493

19.1 Fundamentals ..... 494
   Overview ..... 494
   Differences between Cycles 31 to 33 and Cycles 481 to 483 ..... 495
   Setting the machine parameters ..... 495
   Entries in the tool table TOOL.T ..... 497
   Display of the measurement results ..... 498

19.2 Calibrating the TT (Cycle 30 or 480, DIN/ISO: G480) ..... 499
   Cycle run ..... 499
   Please note while programming: ..... 499
   Cycle parameters ..... 499

19.3 CALIBRATING THE WIRELESS TT 449 (Cycle 484, DIN/ISO: G484) ..... 500
   Fundamentals ..... 500
   Cycle run ..... 500
   Please note while programming: ..... 500
   Cycle parameters ..... 500

19.4 Measuring the Tool Length (Cycle 31 or 481, DIN/ISO: G481) ..... 501
   Cycle run ..... 501
   Please note while programming: ..... 502
   Cycle parameters ..... 502

19.5 Measuring the Tool Radius (Cycle 32 or 482, ISO: G482) ..... 503
   Cycle run ..... 503
   Please note while programming: ..... 503
   Cycle parameters ..... 504

19.6 Measuring Tool Length and Radius (Cycle 33 or 483, ISO: G483) ..... 505
   Cycle run ..... 505
   Please note while programming: ..... 505
   Cycle parameters ..... 506

Overview ..... 509
   Fixed cycles ..... 509
   Touch probe cycles ..... 511
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. Parameters with specific functions that are required in several cycles always have the same number: For example, Q200 is always assigned the setup clearance, Q202 the plunging depth, etc.

\[ \text{Danger of collision!} \]

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.

<table>
<thead>
<tr>
<th>Cycle group</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cycles for pecking, reaming, boring, and counterboring</td>
<td>DRILLING/THREAD</td>
<td>72</td>
</tr>
<tr>
<td>Cycles for tapping, thread cutting and thread milling</td>
<td>DRILLING/THREAD</td>
<td>106</td>
</tr>
<tr>
<td>Cycles for milling pockets, studs and slots</td>
<td>POCKETS/STUDS/SLOTS</td>
<td>140</td>
</tr>
<tr>
<td>Cycles for producing point patterns, such as circular or linear hole patterns</td>
<td>PATTERN</td>
<td>172</td>
</tr>
<tr>
<td>SL (Subcontour List) cycles which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation</td>
<td>SL II</td>
<td>184</td>
</tr>
<tr>
<td>Cycles for multipass milling of flat or twisted surfaces</td>
<td>MULTIPASS MOLLING</td>
<td>254</td>
</tr>
<tr>
<td>Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours</td>
<td>COORD. TRANSF.</td>
<td>272</td>
</tr>
<tr>
<td>Special cycles such as dwell time, program call, oriented spindle stop and tolerance</td>
<td>SPECIAL CYCLES</td>
<td>302</td>
</tr>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Cycle group</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cycles for automatic measurement and compensation of workpiece misalignment</td>
<td></td>
<td>320</td>
</tr>
<tr>
<td>Cycles for automatic workpiece presetting</td>
<td></td>
<td>342</td>
</tr>
<tr>
<td>Cycles for automatic workpiece inspection</td>
<td></td>
<td>396</td>
</tr>
<tr>
<td>Calibration cycles, special cycles</td>
<td></td>
<td>446</td>
</tr>
<tr>
<td>Cycles for automatic kinematics measurement</td>
<td></td>
<td>462</td>
</tr>
<tr>
<td>Cycles for automatic tool measurement (enabled by the machine tool builder)</td>
<td></td>
<td>494</td>
</tr>
</tbody>
</table>

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

Sometimes, machine-specific cycles also use transfer parameters, which HEIDENHAIN already used in the standard cycles. The TNC executes DEF-active cycles as soon as they are defined (see also “Calling cycles” on page 50). It executes CALL-active cycles only after they have been called (see also “Calling cycles” on page 50). When DEF-active cycles and CALL-active cycles are used simultaneously, it is important to prevent overwriting of transfer parameters already in use. Use the following procedure:

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

Refer to your machine manual for a description of the specific function.
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 desired cycle, for example THREAD MILLING. The TNC initiates the programming dialog and asks 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 shows an overview of cycles in a pop-up window
- Choose the desired cycle with the arrow keys, or
- Choose the desired cycle with CTRL and the arrow keys (for pagewise scrolling), or
- Enter the cycle number and confirm it with the ENT key. The TNC then initiates the cycle dialog as described above

Example NC blocks

```
7 CYCL DEF 200 DRILLING
Q200=2 ; SETUP 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 SETUP CLEARANCE
Q211=0.25 ; DWELL TIME AT DEPTH
```
Calling cycles

Prerequisites

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

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

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

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

- Cycle 220 for 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 fixed cycle at all positions that you defined in a PATTERN DEF pattern definition (see “Pattern Definition PATTERN DEF” on page 58) or in a point table (see “Point Tables” on page 66).
2.1 Working with Fixed Cycles

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 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 current tool 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.

As a rule, the TNC moves without radius compensation (R0) to the position defined in the CYCL CALL POS block.

If you use CYCL CALL POS to call a cycle in which a start position is defined (for example Cycle 212), then the position defined in the cycles serves as an additional shift on the position defined in the CYCL CALL POS block. You should therefore always define the start position to be set in the cycle as 0.

Calling a cycle with M99/89

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

If the TNC is to execute the cycle automatically after every positioning block, program the first cycle call with M89 (depending on MP 7440).

To cancel the effect of M89, program:

- M99 in the positioning block in which you move to the last starting point, or
- A CYCL CALL POS block or
- A new fixed cycle with CYCL DEF
Working with the secondary axes U/V/W

The TNC performs infeed movements in the axis that was defined in the TOOL CALL block as the spindle axis. It performs movements in the working plane only in the principal axes X, Y or Z. Exceptions:

- You program secondary axes for the side lengths in cycles 3 SLOT MILLING and 4 POCKET MILLING
- You program secondary axes in the first block of the contour geometry subprogram of an SL cycle
- In Cycles 5 (CIRCULAR POCKET), 251 (RECTANGULAR POCKET), 252 (CIRCULAR POCKET), 253 (SLOT) and 254 (CIRCULAR SLOT), the TNC machines the cycle in the axes that you programmed in the last positioning block before the cycle call. When tool axis Z is active, the following combinations are permissible:
  - X/Y
  - X/V
  - U/Y
  - U/V
### 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 setup clearance \texttt{Q200}, which you must enter for each cycle definition. The \texttt{GLOBAL DEF} function gives you the possibility of defining these cycle parameters once at the beginning of the program, so that they are effective globally for all fixed cycles used in the program. In the respective fixed cycle you then simply link to the value defined at the beginning of the program.

The following \texttt{GLOBAL DEF} functions are available:

<table>
<thead>
<tr>
<th>Machining pattern</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>GLOBAL DEF COMMON</td>
<td></td>
<td>Page 55</td>
</tr>
<tr>
<td>Definition of generally valid cycle parameters</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GLOBAL DEF DRILLING</td>
<td></td>
<td>Page 55</td>
</tr>
<tr>
<td>Definition of specific drilling cycle parameters</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GLOBAL DEF POCKET MILLING</td>
<td></td>
<td>Page 56</td>
</tr>
<tr>
<td>Definition of specific pocket-milling cycle parameters</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GLOBAL DEF CONTOUR MILLING</td>
<td></td>
<td>Page 56</td>
</tr>
<tr>
<td>Definition of specific contour milling parameters</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GLOBAL DEF POSITIONING</td>
<td></td>
<td>Page 56</td>
</tr>
<tr>
<td>Definition of the positioning behavior for \texttt{CYCL CALL PAT}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GLOBAL DEF PROBING</td>
<td></td>
<td>Page 57</td>
</tr>
<tr>
<td>Definition of specific touch probe cycle parameters</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
2.2 Program Defaults for Cycles

Entering GLOBAL DEF

- Select the Programming and Editing operating mode.
- Press the Special Functions key.
- Select the functions for program defaults.
- Select GLOBAL DEF functions.
- Select the desired GLOBAL DEF function, e.g. GLOBAL DEF COMMON
- Enter the required definitions, and confirm each entry with the ENT key.

Using GLOBAL DEF information

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

Proceed as follows:

- Select the Programming and Editing operating mode
- Select fixed cycles
- Select the desired group of cycles, for example: drilling cycles
- Select the desired cycle, e.g. DRILLING
- The TNC displays the SET STANDARD VALUES soft key, if there is a global parameter for it
- Press the SET STANDARD VALUES soft key. The TNC enters the word PREDEF (predefined) in the cycle definition. You have now created a link to the corresponding GLOBAL DEF parameter that you defined at the beginning of the program

Danger of collision!

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

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

- **Setup 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 setup clearance

The parameters apply to the drilling, tapping and thread milling cycles 200 to 209, 240, and 262 to 267.
Global data for milling operations with pocket cycles 25x

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

The parameters apply to milling cycles 251 to 257.

Global data for milling operations with contour cycles

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

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

Global data for positioning behavior

- **Positioning behavior**: Retraction in the tool axis at the end of the machining step: Return to the 2nd setup 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

- **Setup 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 setup clearance or clearance height between the measuring points.

Applies to all Touch Probe Cycles 4xx.
2.3 Pattern Definition PATTERN DEF

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:

<table>
<thead>
<tr>
<th>Machining pattern</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>POINT</td>
<td><img src="image" alt="POINT" /></td>
<td>Page 60</td>
</tr>
<tr>
<td>Definition of up to any 9 machining positions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ROW</td>
<td><img src="image" alt="ROW" /></td>
<td>Page 61</td>
</tr>
<tr>
<td>Definition of a single row, straight or rotated</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PATTERN</td>
<td><img src="image" alt="PATTERN" /></td>
<td>Page 62</td>
</tr>
<tr>
<td>Definition of a single pattern, straight, rotated or distorted</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FRAME</td>
<td><img src="image" alt="FRAME" /></td>
<td>Page 63</td>
</tr>
<tr>
<td>Definition of a single frame, straight, rotated or distorted</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CIRCLE</td>
<td><img src="image" alt="CIRCLE" /></td>
<td>Page 64</td>
</tr>
<tr>
<td>Definition of a full circle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PITCH CIRCLE</td>
<td><img src="image" alt="PITCH CIRCLE" /></td>
<td>Page 65</td>
</tr>
<tr>
<td>Definition of a circular arc</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## Entering PATTERN DEF

- Select the Programming and Editing operating mode
- Press the Special Functions key
- Select the functions for contour and point machining
- Open a PATTERN DEF block
- Select the desired machining pattern, e.g. a single row
- Enter the required definitions, and confirm each entry with the ENT key

## Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the CYCL CALL PAT function (see “Calling a cycle with CYCL CALL PAT” on page 50). The TNC then performs the most recently defined machining cycle on the machining pattern you defined.

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

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

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

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

**Example: NC blocks**

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

- **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 positions:** Total number of machining positions.
- **Rot. position of entire pattern (absolute):** Angle of rotation around the entered starting point. Reference axis: Major axis of the active machining plane (e.g., X for tool axis Z). You can enter a positive or negative value.
- **Workpiece surface coordinate (absolute):** Enter Z coordinate at which machining is to begin.

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.

**Example: NC blocks**

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

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

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

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

**Example: NC blocks**

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

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

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

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

### Example: 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)
```
2.3 Pattern Definition PATTERN DEF

Defining a full circle

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

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.

**Example: NC blocks**

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

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

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

**Example: 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

Function

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

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

Creating a point table

Select the Programming and Editing mode of operation.

Press the PGM MGT key to call the file manager.

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

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

Repeat the process until all desired coordinates have been entered.

With the soft keys X OFF/ON, Y OFF/ON, Z OFF/ON (second soft-key row), you can 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 and Editing mode of operation, select the program for which you want to activate the point table:

1. Press the PGM CALL key to call the function for selecting the point table.
2. Press the POINT TABLE soft key.
3. Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the desired datum table.

Select a point table with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the SEL PATTERN block.

Conclude this function with the END key.

Alternatively you can also enter the table name or the complete path name of the table to be called directly via the keyboard.

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** not valid)
- If required, enter a miscellaneous function M, then confirm with the END key

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

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

**Effect of the point tables with SL cycles and Cycle 12**
The TNC interprets the points as an additional datum shift.

**Effect of the point tables with Cycles 200 to 208 and 262 to 267**
The TNC interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.

**Effect of the point tables with Cycles 210 to 215**
The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0.

**Effect of the point tables with Cycles 251 to 254**
The TNC interprets the points of the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.
3

Fixed Cycles: Drilling
### 3.1 Basics

#### Overview

The TNC offers 9 cycles for all types of drilling operations:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>240 CENTERING</td>
<td>![240]</td>
<td>Page 73</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth</td>
<td></td>
<td></td>
</tr>
<tr>
<td>200 DRILLING</td>
<td>![200]</td>
<td>Page 75</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>201 REAMING</td>
<td>![201]</td>
<td>Page 77</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>202 BORING</td>
<td>![202]</td>
<td>Page 79</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>203 UNIVERSAL DRILLING</td>
<td>![203]</td>
<td>Page 83</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance, chip breaking, and decrementing</td>
<td></td>
<td></td>
</tr>
<tr>
<td>204 BACK BORING</td>
<td>![204]</td>
<td>Page 87</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>205 UNIVERSAL PECKING</td>
<td>![205]</td>
<td>Page 91</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance, chip breaking, and advanced stop distance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>208 BORE MILLING</td>
<td>![208]</td>
<td>Page 95</td>
</tr>
<tr>
<td>With automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>241 SINGLE-LIP DEEP-HOLE DRILLING</td>
<td>![241]</td>
<td>Page 98</td>
</tr>
<tr>
<td>With automatic pre-positioning to deepened starting point, shaft speed and coolant definition</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3.2 CENTERING (Cycle 240, DIN/ISO: G240)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse $F_{\text{MAX}}$ to the setup clearance above the workpiece surface.
2. The tool is centered at the programmed feed rate $F$ to the entered centering diameter or centering depth.
3. If defined, the tool remains at the centering depth.
4. Finally, the tool moves to setup clearance or—if programmed—to the 2nd setup clearance at rapid traverse $F_{\text{MAX}}$.

Please note while programming:

- **Program a positioning block for the starting point (hole center) in the working plane with radius compensation $R_0$.**
  - The algebraic sign for the cycle parameter $Q_{344}$ (diameter) or $Q_{201}$ (depth) determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

- **Danger of collision!**
  - Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.
  - 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 setup clearance **below** the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999, alternatively PREDEF

- **Select Depth/Diameter (1/0)** Q343: Select whether centering is based on the entered diameter or depth. If the TNC is to center based on the entered diameter, the point angle of the tool must be defined in the T-ANGLE column of the tool table TOOL.T.
  - 0: Centering based on the entered depth
  - 1: Centering based on the entered diameter

- **Depth** Q201 (incremental value): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range −99999.9999 to 99999.9999

- **Diameter (algebraic sign)** Q344: Centering diameter. Only effective if Q343=1 is defined. Input range −99999.9999 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool during centering in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU.

- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively PREDEF

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Example: NC blocks

```
 10 L Z+100 R0 FMAX
 11 CYCL DEF 240 CENTERING
    Q200=2 ;SETUP CLEARANCE
    Q343=1 ;SELECT DEPTH/DIA.
    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 SETUP CLEARANCE
 12 CYCL CALL POS X+30 Y+20 Z+0 FMAX M3
 13 CYCL CALL POS X+80 Y+50 Z+0 FMAX
```
3.3 DRILLING (Cycle 200)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse $F_{\text{MAX}}$ to the setup 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 $F_{\text{MAX}}$ to the setup clearance, dwells there (if a dwell time was entered), and then moves at $F_{\text{MAX}}$ to the setup clearance above the first plunging depth.

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

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

6. The tool is retracted from the hole bottom to the setup clearance or—if programmed—to the 2nd setup clearance at $F_{\text{MAX}}$.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation $R_0$.

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

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999, alternatively PREDEF

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU.

- **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - The plunging depth is equal to the depth
  - The plunging depth is greater than the depth

- **Dwell time at top** Q210: Time in seconds that the tool remains at setup clearance after having been retracted from the hole for chip release. Input range 0 to 3600.0000, alternatively PREDEF

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively PREDEF

Example: NC blocks

```
11 CYCL DEF 200 DRILLING
Q200=2 ;SETUP CLEARANCE
Q201=-15 ;DEPTH
Q206=250 ;FEED RATE FOR PLNGNG
Q202=5 ;PLUNGING DEPTH
Q210=0 ;DWELL TIME AT TOP
Q203=+20 ;SURFACE COORDINATE
Q204=100 ;2ND SETUP CLEARANCE
Q211=0.1 ;DWELL TIME AT DEPTH
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 spindle axis at rapid traverse $F_{\text{MAX}}$ to the entered setup 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 the setup clearance at the feed rate $F$, and from there—if programmed—to the 2nd setup clearance at $F_{\text{MAX}}$.

Please note while programming:

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation $R_0$.
- 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!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool during reaming in mm/min. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**.

- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively **PREDEF**

- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the reaming feed rate. Input range 0 to 99999.9999

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**

Example: NC blocks

```
11 CYCL DEF 201 REAMING
    Q200=2 ;SETUP CLEARANCE
    Q201=-15 ;DEPTH
    Q206=100 ;FEED RATE FOR PLNGNG
    Q211=0.5 ;D Well TIME AT DEPTH
    Q208=250 ;RETRACTION FEED RATE
    Q203=+20 ;SURFACE COORDINATE
    Q204=100 ;2ND SETUP 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 spindle axis at rapid traverse $F_{\text{MAX}}$ to the setup 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 TNC moves the tool at the retraction feed rate to the setup clearance and then, if entered, to the 2nd setup clearance at $F_{\text{MAX}}$. If Q214=0, the tool point remains on the wall of the hole.
Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.
This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.
After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

Danger of collision!
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Select a disengaging direction in which the tool moves away from the edge of the hole.
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.
During retraction the TNC automatically takes an active rotation of the coordinate system into account.
Cycle parameters

- **Setup clearance Q200** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Depth Q201** (incremental): Distance between workpiece surface and bottom of hole. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging Q206**: Traversing speed of the tool during boring at mm/min. Input range: 0 to 99999.9999; alternatively **FAUTO, FU**.

- **Dwell time at depth Q211**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively **PREDEF**

- **Retraction feed rate Q208**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at feed rate for plunging. Input range 0 to 99999.9999, alternatively **FMAX, FAUTO, PREDEF**

- **Workpiece surface coordinate Q203** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance Q204** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**
Disengaging direction (0/1/2/3/4) Q214:
Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation).

0  Do not retract tool.
1  Retract tool in the negative ref. axis direction.
2  Retract tool in the negative minor axis direction.
3  Retract tool in the positive ref. axis direction.
4  Retract tool in the positive minor axis direction.

Angle for spindle orientation Q336 (absolute):
Angle at which the TNC positions the tool before retracting it. Input range -360.000 to 360.000

Example:

```
10 L Z+100 R0 FMAX
11 CYCL DEF 202 BORING
   Q200=2 ;SETUP 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 SETUP CLEARANCE
   Q214=1 ;DISENGAGING DIRECTN
   Q336=0 ;ANGLE OF SPINDLE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
```
3.6 UNIVERSAL DRILLING  
(Cycle 203, DIN/ISO: G203)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse \( F_{\text{MAX}} \) to the entered setup clearance above the workpiece surface.

2. The tool drills to the first plunging depth at the programmed 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 setup clearance, remains there—if programmed—for the entered dwell time, and advances again at \( F_{\text{MAX}} \) to the setup 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 the setup clearance at the retraction feed rate. If programmed, the tool moves to the 2nd setup clearance at \( F_{\text{MAX}} \).
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!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range: 0 to 99999.9999; alternatively **FAUTO, FU**.

- **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth and no chip breaking is defined

- **Dwell time at top** Q210: Time in seconds that the tool remains at setup clearance after having been retracted from the hole for chip release. Input range 0 to 3600.0000, alternatively **PREDEF**

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999
Fixed Cycles: Drilling 3.6 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)

- **No. of breaks before retracting** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip release. For chip breaking, the TNC retracts the tool each time by the value in Q256. Input range 0 to 99999.

- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999.

- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively PREDEF.

- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q206. Input range 0 to 99999.9999, alternatively FMAX, FAUTO, PREDEF.

- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.1000 to 99999.9999, alternatively PREDEF.

Example: NC blocks

```
11 CYCL DEF 203 UNIVERSAL DRILLING
Q200=2 ;SETUP CLEARANCE
Q201=-20 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q202=5 ;PLUNGING DEPTH
Q210=0 ;DWELL TIME AT TOP
Q203=+20 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q212=0.2 ;DECREMENT
Q213=3 ;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
```
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 spindle axis at rapid traverse $F_{\text{MAX}}$ to the setup 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 the setup 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. Another oriented spindle stop is carried out and the tool is once again displaced by the off-center distance.

6. The TNC moves the tool at the pre-positioning feed rate to the setup clearance and then—if entered—to the 2nd setup clearance at $F_{\text{MAX}}$. 
Please note while programming:

- Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.
- This cycle is effective only for machines with servo-controlled spindle.
- Special boring bars for upward cutting are required for this cycle.

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

- The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.
- The entered tool length is the total length to the underside of the boring bar and not just to the tooth.
- When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.
- You can also execute Cycle 204 with M04 if you have programmed M04 instead of M03 prior to the cycle call.

Danger of collision!

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

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF.

- **Depth of counterbore** Q249 (incremental): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range -99999.9999 to 99999.9999.

- **Material thickness** Q250 (incremental): Thickness of the workpiece. Input range 0.0001 to 99999.9999.

- **Off-center distance** Q251 (incremental): Off-center distance for the boring bar; value from tool data sheet. Input range 0.0001 to 99999.9999.

- **Tool edge height** Q252 (incremental): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet. Input range 0.0001 to 99999.9999.

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively FMAX, FAUTO, PREDEF.

- **Feed rate for back boring** Q254: Traversing speed of the tool during back boring in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU.

- **Dwell time** Q255: Dwell time in seconds at the top of the bore hole. Input range 0 to 3600.000.
Fixed Cycles: Drilling

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

- **Workpiece surface coordinate Q203 (absolute):** Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

- **2nd setup clearance Q204 (incremental):** Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

- **Disengaging direction (0/1/2/3/4) Q214:** Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation). Input of 0 is not permitted.
  1. Retract tool in the negative ref. axis direction.
  2. Retract tool in the negative minor axis direction.
  3. Retract tool in the positive ref. axis direction.
  4. Retract tool in the positive minor axis direction.

- **Angle for spindle orientation Q336 (absolute):** Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole. Input range -360.0000 to 360.0000

**Example: NC blocks**

```
11 CYCL DEF 204 BACK BORING
Q200=2 ;SETUP CLEARANCE
Q249=+5 ;DEPTH OF COUNTERBORE
Q250=20 ;MATERIAL THICKNESS
Q251=3.5 ;OFF-CENTER DISTANCE
Q252=15 ;TOOL EDGE HEIGHT
Q253=750 ;F PRE-POSITIONING
Q254=200 ;F COUNTERSINKING
Q255=0 ;DWELL TIME
Q203=+20 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP 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 $F_{\text{MAX}}$ to the programmed setup clearance above the workpiece surface.

2. If you enter a deepened starting point, the TNC moves at the defined positioning feed rate to the setup clearance above the deepened starting point.

3. The tool drills to the first plunging depth at the programmed 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 setup clearance, and then at $F_{\text{MAX}}$ 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 the setup clearance at the retraction feed rate. If programmed, the tool moves to the 2nd setup clearance at $F_{\text{MAX}}$. 
Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation \( R_0 \).

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 advance stop distances \( Q_{258} \) not equal to \( Q_{259} \), the TNC will change the advance stop distances between the first and last plunging depths at the same rate.

If you use \( Q_{379} \) to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. Retraction movements are not changed by the TNC, therefore they are calculated with respect to the coordinate of the workpiece surface.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** \(Q200\) (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Depth** \(Q201\) (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999

- **Feed rate for plunging** \(Q206\): Traversing speed of the tool during drilling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU.

- **Plunging depth** \(Q202\) (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

- **Workpiece surface coordinate** \(Q203\) (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** \(Q204\) (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Decrement** \(Q212\) (incremental): Value by which the TNC decreases the plunging depth \(Q202\). Input range 0 to 99999.9999

- **Minimum plunging depth** \(Q205\) (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with \(Q205\). Input range 0 to 99999.9999

- **Upper advanced stop distance** \(Q258\) (incremental): Setup clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth. Input range 0 to 99999.9999

- **Lower advanced stop distance** \(Q259\) (incremental): Setup clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth. Input range 0 to 99999.9999
Infeed depth for chip breaking Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999

Retraction rate for chip breaking Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. The TNC retracts the tool at a feed rate of 3000 mm/min. Input range 0.1000 to 99999.9999, alternatively PREDEF.

Dwell time at depth Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively PREDEF

Deepened starting point Q379 (incremental with respect to the workpiece surface): Starting position of drilling if a shorter tool has already pilot drilled to a certain depth. The TNC moves at the feed rate for pre-positioning from the setup clearance to the deepened starting point. Input range 0 to 99999.9999

Feed rate for pre-positioning Q253: Traversing velocity of the tool during positioning from the setup clearance to a deepened starting point in mm/min. Effective only if Q379 is entered not equal to 0. Input range 0 to 99999.9999, alternatively FMAX, FAUTO, PREDEF

Example: NC blocks

```
11 CYCL DEF 205 UNIVERSAL PECKING
Q200=2 ;SETUP CLEARANCE
Q201=-80 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q202=15 ;PLUNGING DEPTH
Q203=+100 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP 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
```
3.9 BORE MILLING (Cycle 208)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse $F_{\text{MAX}}$ to the programmed setup 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 $F_{\text{MAX}}$. If programmed, the tool moves to the 2nd setup clearance at $F_{\text{MAX}}$. 
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 MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool lower edge and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool during helical drilling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

- **Infeed per helix** Q334 (incremental): Depth of the tool plunge with each helix (=360°). Input range 0 to 99999.9999

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

- **Nominal diameter** Q335 (absolute value): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation. Input range 0 to 99999.9999

- **Roughing diameter** Q342 (absolute): As soon as you enter a value greater than 0 in Q342, the TNC no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter. Input range 0 to 99999.9999

- **Climb or up-cut** Q351: Type of milling operation with M3
  - +1 = climb milling
  - -1 = up-cut milling
  - PREDEF = use the default value from GLOBAL DEF

---

**Example: NC blocks**

```
12 CYCL DEF 208 BORE MILLING
Q200=2 ;SETUP CLEARANCE
Q201=-80 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q334=1.5 ;PLUNGING DEPTH
Q203=+100 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q335=25 ;NOMINAL DIAMETER
Q342=0 ;ROUGHING DIAMETER
Q351=+1 ;CLIMB OR UP-CUT
```
3.10 SINGLE-FLUTED DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse $F_{MAX}$ to the entered setup clearance above the workpiece surface.

2 Then the TNC moves the tool at the defined positioning feed rate to the setup clearance above the deepened starting point and switches on the drilling speed (M3) and the coolant. The approach motion is executed at the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.

3 The tool drills to the entered drilling depth or, if defined so, to the entered dwell depth, at the programmed feed rate $F$.

4 If programmed, the tool remains at the hole bottom for chip breaking. Then the TNC switches off the coolant and resets the drilling speed to the value defined for retraction.

5 After the dwell time at the hole bottom, the tool is retracted to the setup clearance at the retraction feed rate. If programmed, the tool moves to the 2nd setup clearance at $F_{MAX}$.

Please note while programming:

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation $R_0$.
- 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!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance **below** the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range: -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000, alternatively PREDEF
- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF
- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position for actual drilling operation. The TNC moves at the feed rate for pre-positioning from the setup clearance to the deepened starting point. Input range 0 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing velocity of the tool during positioning from the setup clearance to the deepened starting point in mm/min. Effective only if Q379 is entered not equal to 0. Input range 0 to 99999.9999, alternatively FMAX, FAUTO, PREDEF
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q206. Input range 0 to 99999.9999, alternatively FMAX, FAUTO, PREDEF
Rotat. dir. of entry/exit (3/4/5) Q426: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input range:
3: Spindle rotation with M3
4: Spindle rotation with M4
5: Movement with stationary spindle

Spindle speed of entry/exit Q427: Desired spindle speed when tool moves into and retracts from the hole. Input range 0 to 99999

Drilling speed Q428: Desired speed for drilling. Input range 0 to 99999

M function for coolant on? Q429: M function for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at the deepened starting point. Input range 0 to 999

M function for coolant off? Q430: M function for switching off the coolant. The TNC switches the coolant off if the tool is at the hole depth. Input range 0 to 999

Dwell depth Q435 (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (standard setting). Application: During machining of through-holes some tools require a short dwell time before exiting the bottom of the hole in order to transport the chips to the top. Define a value smaller than the hole depth Q201; input range 0 to 99999.9999.

Example: NC blocks

```
11 CYCL DEF 241 SINGLE-LIP DEEP-HOLE DRILLING
Q200=2 ;SETUP CLEARANCE
Q201=-80 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q211=0.25 ;DWELL TIME AT DEPTH
Q203=+100 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP 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 ;DRILLING SPEED
Q429=8 ;COOLANT ON
Q430=9 ;COOLANT OFF
Q435=0 ;DWELL DEPTH
```
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 ;SETUP CLEARANCE
   Q201=-15 ;DEPTH
   Q206=250 ;FEED RATE FOR PLNGN
   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
### 3.11 Programming Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>L X+10 Y+10 R0 FMAX M3</td>
<td>Approach hole 1, spindle ON</td>
</tr>
<tr>
<td>7</td>
<td>CYCL CALL</td>
<td>Cycle call</td>
</tr>
<tr>
<td>8</td>
<td>L Y+90 R0 FMAX M99</td>
<td>Approach hole 2, call cycle</td>
</tr>
<tr>
<td>9</td>
<td>L X+90 R0 FMAX M99</td>
<td>Approach hole 3, call cycle</td>
</tr>
<tr>
<td>10</td>
<td>L Y+10 R0 FMAX M99</td>
<td>Approach hole 4, call cycle</td>
</tr>
<tr>
<td>11</td>
<td>L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>12</td>
<td>END PGM C200 MM</td>
<td></td>
</tr>
</tbody>
</table>
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)

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM 1 MM</td>
</tr>
<tr>
<td>1</td>
<td>BLK FORM 0.1 Z X+0 Y+0 Z-20</td>
</tr>
<tr>
<td>2</td>
<td>BLK FORM 0.2 X+100 Y+100 Y+0</td>
</tr>
<tr>
<td>3</td>
<td>TOOL CALL 1 Z S5000</td>
</tr>
<tr>
<td>4</td>
<td>L Z+10 R0 F5000</td>
</tr>
<tr>
<td>5</td>
<td>PATTERN DEF</td>
</tr>
<tr>
<td></td>
<td>POS1( X+10 Y+10 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS2( X+40 Y+30 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS3( X+20 Y+55 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS4( X+10 Y+90 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS5( X+90 Y+90 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS6( X+80 Y+65 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS7( X+80 Y+30 Z+0 )</td>
</tr>
<tr>
<td></td>
<td>POS8( X+90 Y+10 Z+0 )</td>
</tr>
</tbody>
</table>
### 6 CYCL DEF 240 CENTERING

<table>
<thead>
<tr>
<th>Q200=2</th>
<th>;SETUP CLEARANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q343=0</td>
<td>;SELECT DEPTH/DIA.</td>
</tr>
<tr>
<td>Q201=-2</td>
<td>;DEPTH</td>
</tr>
<tr>
<td>Q344=-10</td>
<td>;DIAMETER</td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PLNGN</td>
</tr>
<tr>
<td>Q211=0</td>
<td>;DWELL TIME AT DEPTH</td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SETUP CLEARANCE</td>
</tr>
</tbody>
</table>

**Cycle definition: CENTERING**

- **Q200=2**: Setup Clearance
- **Q343=0**: Select Depth/Diameter
- **Q201=-2**: Depth
- **Q344=-10**: Diameter
- **Q206=150**: Feed Rate for Plunging
- **Q211=0**: Dwell Time at Depth
- **Q203=+0**: Surface Coordinate
- **Q204=50**: 2nd Setup Clearance

### 7 CYCL CALL PAT F5000 M13

Call the cycle in connection with point pattern

### 8 L Z+100 R0 FMAX

Retract the tool, change the tool

### 9 TOOL CALL 2 Z S5000

Call the drilling tool (radius 2.4)

### 10 L Z+10 R0 F5000

Move tool to clearance height (enter a value for F)

### 11 CYCL DEF 200 DRILLING

**Cycle definition: drilling**

<table>
<thead>
<tr>
<th>Q200=2</th>
<th>;SETUP CLEARANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q201=-25</td>
<td>;DEPTH</td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PECKING</td>
</tr>
<tr>
<td>Q202=5</td>
<td>;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q210=0</td>
<td>;DWELL TIME AT TOP</td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q211=0.2</td>
<td>;DWELL TIME AT DEPTH</td>
</tr>
</tbody>
</table>

### 12 CYCL CALL PAT F5000 M13

Call the cycle in connection with point pattern

### 13 L Z+100 R0 FMAX

Retract the tool

### 14 TOOL CALL 3 Z S200

Call the tapping tool (radius 3)

### 15 L Z+50 R0 FMAX

Move tool to clearance height

### 16 CYCL DEF 206 TAPPING NEW

**Cycle definition for tapping**

<table>
<thead>
<tr>
<th>Q200=2</th>
<th>;SETUP CLEARANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q201=-25</td>
<td>;DEPTH OF THREAD</td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PECKING</td>
</tr>
<tr>
<td>Q211=0</td>
<td>;DWELL TIME AT DEPTH</td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SETUP CLEARANCE</td>
</tr>
</tbody>
</table>

### 17 CYCL CALL PAT F5000 M13

Call the cycle in connection with point pattern

### 18 L Z+100 R0 FMAX M2

Retract in the tool axis, end program

### 19 END PGM 1 MM

---

104  Fixed Cycles: Drilling
Fixed Cycles: Tapping / Thread Milling
4.1 Basics

Overview

The TNC offers 8 cycles for all types of threading operations:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>206 TAPPING NEW</td>
<td>Page 107</td>
<td></td>
</tr>
<tr>
<td>With a floating tap holder, with automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>207 RIGID TAPPING NEW</td>
<td>Page 109</td>
<td></td>
</tr>
<tr>
<td>Without a floating tap holder, with automatic pre-positioning, 2nd setup clearance</td>
<td></td>
<td></td>
</tr>
<tr>
<td>209 TAPPING W/ CHIP BREAKING</td>
<td>Page 112</td>
<td></td>
</tr>
<tr>
<td>Without a floating tap holder, with automatic pre-positioning, 2nd setup clearance, chip breaking</td>
<td></td>
<td></td>
</tr>
<tr>
<td>262 THREAD MILLING</td>
<td>Page 117</td>
<td></td>
</tr>
<tr>
<td>Cycle for milling a thread in pre-drilled material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>263 THREAD MILLING/CNTSNKG</td>
<td>Page 120</td>
<td></td>
</tr>
<tr>
<td>Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer</td>
<td></td>
<td></td>
</tr>
<tr>
<td>264 THREAD DRILLING/MILLING</td>
<td>Page 124</td>
<td></td>
</tr>
<tr>
<td>Cycle for drilling into the solid material with subsequent milling of the thread with a tool</td>
<td></td>
<td></td>
</tr>
<tr>
<td>265 HEL.THREAD DRILLING/MILLING</td>
<td>Page 128</td>
<td></td>
</tr>
<tr>
<td>Cycle for milling the thread into the solid material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>267 OUTSIDE THREAD MILLING</td>
<td>Page 128</td>
<td></td>
</tr>
<tr>
<td>Cycle for milling an external thread and machining a countersunk chamfer</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
4.2 TAPPING NEW with a Floating Tap Holder (Cycle 206, DIN/ISO: G206)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse $F_{\text{MAX}}$ to the entered setup 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 setup clearance at the end of the dwell time. If programmed, the tool moves to the 2nd setup clearance at $F_{\text{MAX}}$.
4. At the setup clearance, the direction of spindle rotation reverses once again.

Please note while programming:

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation $R_0$.
- 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.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch. Input range 0 to 99999.9999, alternatively PREDEF

- **Total hole depth** Q201 (thread length, incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999

- **Feed rate** F Q206: Traversing speed of the tool during tapping. Input range: 0 to 99999.9999, alternatively FAUTO

- **Dwell time at bottom** Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000, alternatively PREDEF

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

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.

Example: NC blocks

```
25 CYCL DEF 206 TAPPING NEW
Q200=2 ;SETUP CLEARANCE
Q201=-20 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q211=0.25 ;DWELL TIME AT DEPTH
Q203=+25 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
```
4.3 RIGID TAPPING without a Floating Tap Holder NEW (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 spindle axis at rapid traverse \( F_{\text{MAX}} \) to the entered setup 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 setup clearance at the end of the dwell time. If programmed, the tool moves to the 2nd setup clearance at \( F_{\text{MAX}} \).
4. The TNC stops the spindle turning at setup clearance.
Please note while programming:

- Machine and TNC must be specially prepared by the machine tool builder for use of this cycle. This cycle is effective only for machines with servo-controlled spindle.

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.
  - The algebraic sign for the total hole depth parameter determines the working direction.
  - The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.
  - The feed-rate override knob is disabled.
  - At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

**Danger of collision!**

- Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

- **Total hole depth** Q201 (incremental): Distance between workpiece surface and end of thread. Input range: -99999.9999 to 99999.9999

- **Pitch** Q239
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  
  + = right-hand thread
  
  − = left-hand thread
  
  Input range -99.9999 to 99.9999

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active spindle axis.

Example: NC blocks

```
26 CYCL DEF 207 RIGID TAPPING NEW
Q200=2 ;SETUP CLEARANCE
Q201=-20 ;DEPTH
Q239=+1 ;PITCH
Q203=+25 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
```
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 \( F_{MAX} \) to the programmed setup 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 breaking, 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 the setup clearance. If programmed, the tool moves to the 2nd setup clearance at \( F_{MAX} \).

6. The TNC stops the spindle turning at setup clearance.
Please note while programming:

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

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

Program a positioning block for the starting point (hole center) in the working plane with radius compensation \( R_0 \).

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

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

The feed-rate override knob is disabled.

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

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

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
Cycle parameters

- **Setup clearance** \( Q200 \) (incremental): Distance between tool tip (at starting position) and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Thread depth** \( Q201 \) (incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999

- **Pitch** \( Q239 \)
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  
  \(+\) = right-hand thread
  
  \(-\) = left-hand thread
  
  Input range -99.9999 to 99.9999

- **Workpiece surface coordinate** \( Q203 \) (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

- **2nd setup clearance** \( Q204 \) (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Infeed depth for chip breaking** \( Q257 \) (incremental): Depth at which TNC carries out chip breaking. Input range 0 to 99999.9999

- **Retraction rate for chip breaking** \( Q256 \): The TNC multiplies the pitch \( Q239 \) by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter \( Q256 = 0 \), the TNC retracts the tool completely from the hole (to the setup clearance) for chip breaking. Input range 0.1000 to 99999.9999

- **Angle for spindle orientation** \( Q336 \) (absolute): Angle at which the TNC positions the tool before machining the thread. This allows you to regroove the thread, if required. Input range -360.0000 to 360.0000.

- **RPM factor for retraction** \( Q403 \): Factor by which the TNC increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole. Input range 0.0001 to 10, rpm is increased at most to the maximum speed of the active gear range.

**Retracting after a program interruption**

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active spindle axis.

---

**Example: NC blocks**

```
26 CYCL DEF 209 TAPPING W/ CHIP BRKG
Q200=2 ;SETUP CLEARANCE
Q201=-20 ;DEPTH
Q239=+1 ;PITCH
Q203=+25 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q257=5 ;DEPTH FOR CHIP BRKNG
Q256=+25 ;DIST. FOR CHIP BRKNG
Q336=50 ;ANGLE OF SPINDLE
Q403=1.5 ;RPM FACTOR
```
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 $Q_{239}$ (+ = right-hand thread / – = left-hand thread) and milling method $Q_{351}$ (+1 = climb / –1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

<table>
<thead>
<tr>
<th>Internal thread</th>
<th>Pitch</th>
<th>Climb/Up-cut</th>
<th>Work direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right-handed</td>
<td>+</td>
<td>+1(RL)</td>
<td>Z+</td>
</tr>
<tr>
<td>Left-handed</td>
<td>–</td>
<td>–1(RR)</td>
<td>Z+</td>
</tr>
<tr>
<td>Right-handed</td>
<td>+</td>
<td>–1(RR)</td>
<td>Z–</td>
</tr>
<tr>
<td>Left-handed</td>
<td>–</td>
<td>+1(RL)</td>
<td>Z–</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>External thread</th>
<th>Pitch</th>
<th>Climb/Up-cut</th>
<th>Work direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right-handed</td>
<td>+</td>
<td>+1(RL)</td>
<td>Z–</td>
</tr>
<tr>
<td>Left-handed</td>
<td>–</td>
<td>–1(RR)</td>
<td>Z–</td>
</tr>
<tr>
<td>Right-handed</td>
<td>+</td>
<td>–1(RR)</td>
<td>Z+</td>
</tr>
<tr>
<td>Left-handed</td>
<td>–</td>
<td>+1(RL)</td>
<td>Z+</td>
</tr>
</tbody>
</table>

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 MIRRORING 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 the program run, change to the Positioning with MDI operating mode and move the tool in 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 spindle axis at rapid traverse FMAX to the entered setup 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 approaches the thread diameter tangentially in a helical movement. 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 movements or in one continuous movement.

5. After this, 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 at rapid traverse to the setup clearance, or—if programmed—to the 2nd setup clearance.
Please note while programming:

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

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

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

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

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

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
4.6 THREAD MILLING (Cycle 262, DIN/ISO: G262)

Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - - = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Threads per step** Q355: Number of thread revolutions by which the tool is moved:
  - 0 = one 360° helical line to the thread depth
  - 1 = continuous helical path over the entire length of the thread
  - >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch. Input range 0 to 99999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively FMAX, FAUTO, PREDEF.
- **Climb or up-cut** Q351: Type of milling operation with M3
  - +1 = climb milling
  - -1 = up-cut milling
  Alternatively PREDEF
- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999 alternatively PREDEF
- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF
- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999, alternatively FAUTO.

Example: NC blocks

```
25 CYCL DEF 262 THREAD MILLING
Q335=10 ;NOMINAL DIAMETER
Q239=+1.5 ;PITCH
Q201=-20 ;DEPTH OF THREAD
Q355=0 ;THREADS PER STEP
Q253=750 ;F PRE-POSITIONING
Q351=+1 ;CLIMB OR UP-CUT
Q200=2 ;SETUP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q207=500 ;FEED RATE FOR MILLING
```
4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse $F_{\text{MAX}}$ to the entered setup clearance above the workpiece surface.

Countersinking

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

3. If a setup 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 countersinking 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 TNC 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 $360^\circ$ helical motion.

10. After this, 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 at rapid traverse to the setup clearance, or—if programmed—to the 2nd setup clearance.
Please note while programming:

**Before programming, note the following:**

Program a positioning block for the starting point (hole center) in the working plane with radius compensation \( R_0 \).

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:

1st: Depth of thread
2nd: Countersinking depth
3rd: Depth at front

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

If you want to countersink with the front of the tool, 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 MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to setup clearance **below** the workpiece surface!
4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263)

**Cycle parameters**

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Countersinking depth** Q356 (incremental): Distance between tool point and the top surface of the workpiece. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively FMAX, FAUTO, PREDEF.
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling
  Alternatively PREDEF
- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF
- **Setup clearance to the side** Q357 (incremental): Distance between tool tooth and the wall of the hole. Input range 0 to 99999.9999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- Workpiece surface coordinate Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- 2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF
- Feed rate for countersinking Q254: Traversing speed of the tool during countersinking in mm/min. Input range: 0 to 99999.9999, alternatively FAUTO, FU.
- Feed rate for milling Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999, alternatively FAUTO.

Example: NC blocks

```
25 CYCL DEF 263 THREAD MLLNG/CNTSNKG
Q335=10 ;NOMINAL DIAMETER
Q239=+1.5 ;PITCH
Q201=-16 ;DEPTH OF THREAD
Q356=-20 ;COUNTERSINKING DEPTH
Q253=750 ;F PRE-POSITIONING
Q351=+1 ;CLIMB OR UP-CUT
Q200=2 ;SETUP CLEARANCE
Q357=0.2 ;CLEARANCE TO SIDE
Q358=+0 ;DEPTH AT FRONT
Q359=+0 ;OFFSET AT FRONT
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q254=150 ;F COUNTERSINKING
Q207=500 ;FEED RATE FOR MILLING
```
4.8 THREAD DRILLING/MILLING
(Cycle 264, DIN/ISO: G264)

Cycle run

1  The TNC positions the tool in the spindle axis at rapid traverse \( F_{\text{MAX}} \)
to the entered setup 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 setup
clearance, and then at \( F_{\text{MAX}} \) 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
countersinking 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 TNC 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 this, 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 at rapid
traverse to the setup clearance, or—if programmed—to the
2nd setup clearance.
Please note while programming:

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

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Total hole depth
3rd: 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 MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for prepositioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis to setup clearance below the workpiece surface!
4.8 THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264)

**Cycle parameters**

- **Nominal diameter** \( Q335 \): Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** \( Q239 \): Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - – = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** \( Q201 \) (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Total hole depth** \( Q356 \) (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** \( Q253 \): Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively FMAX, FAUTO, PREDEF.
- **Climb or up-cut** \( Q351 \): Type of milling operation with M3
  - +1 = climb milling
  - –1 = up-cut milling
  Alternatively PREDEF
- **Plunging depth** \( Q202 \) (incremental): Infeed per cut. The depth does not have to be a multiple of the plunging depth. Input range 0 to 99999.9999. The TNC will go to depth in one movement if:
  - The plunging depth is equal to the depth
  - The plunging depth is greater than the depth
- **Upper advanced stop distance** \( Q258 \) (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
- **Infeed depth for chip breaking** \( Q257 \) (incremental): Depth at which TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999, alternatively PREDEF
- **Retraction rate for chip breaking** \( Q256 \) (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.1000 to 99999.9999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999.

- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999.

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999.

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**.

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999, alternatively **FAUTO**.

**Example: NC blocks**

```plaintext
25 CYCL DEF 264 THREAD DRILLNG/MLLNG
Q335=10 ;NOMINAL DIAMETER
Q239=+1.5 ;PITCH
Q201=-16 ;DEPTH OF THREAD
Q356=-20 ;TOTAL HOLE DEPTH
Q253=750 ;F PRE-POSITIONING
Q351=+1 ;CLIMB OR UP-CUT
Q202=5 ;PLUNGING DEPTH
Q258=0.2 ;ADVANCED STOP DISTANCE
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 ;SETUP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q206=150 ;FEED RATE FOR PLNGNG
Q207=500 ;FEED RATE FOR MILLING
```
4.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse \( F_{\text{MAX}} \) to the entered setup clearance above the workpiece surface.

Countersinking at front

2. If countersinking is 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 TNC then moves in a semicircle to the hole center.

Thread milling

5. The TNC moves the tool 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 this, 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 at rapid traverse to the setup clearance, or—if programmed—to the 2nd setup clearance.
Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation $R_0$.

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:
1st: Depth of thread
2nd: 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 MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance **below** the workpiece surface!
Cycle parameters

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

- **2nd setup clearance Q204 (incremental):** Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

- **Feed rate for countersinking Q254:** Traversing speed of the tool during countersinking in mm/min. Input range: 0 to 99999.999, alternatively FAUTO, FU.

- **Feed rate for milling Q207:** Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999, alternatively FAUTO.

---

**Example: NC blocks**

```plaintext
25 CYCL DEF 265 HEL. THREAD DRLG/MLG
Q335=10 ;NOMINAL DIAMETER
Q239=+1.5 ;PITCH
Q201=-16 ;DEPTH OF THREAD
Q253=750 ;F PRE-POSITIONING
Q358=+0 ;DEPTH AT FRONT
Q359=+0 ;OFFSET AT FRONT
Q360=0 ;COUNTERSINK
Q200=2 ;SETUP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q254=150 ;F COUNTERSINKING
Q207=500 ;FEED RATE FOR MILLING
```
4.10 OUTSIDE THREAD MILLING
(Cycle 267, DIN/ISO: G267)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse \( F_{\text{MAX}} \) to the entered setup clearance above the workpiece surface.

Countersinking at front

2. The TNC moves in 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 countersinking 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 on 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 movements or in one continuous movement.

10. After this, 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 at rapid traverse to the setup clearance, or—if programmed—to the 2nd setup clearance.
Please note while programming:

Program a positioning block for the starting point (stud center) in the working plane with radius compensation \( R_0 \).

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:
1st: Depth of thread
2nd: 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 MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267)

Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999

- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - - = left-hand thread
  Input range -99.9999 to 99.9999

- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread.

- **Threads per step** Q355: Number of thread revolutions by which the tool is moved:
  - 0 = one helical line to the thread depth
  - 1 = continuous helical path over the entire length of the thread
  - >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch. Input range 0 to 99999

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999 alternatively FMAX, FAUTO, PREDEF.

- **Climb or up-cut** Q351: Type of milling operation with M3
  - +1 = climb milling
  - -1 = up-cut milling
  Alternatively PREDEF
4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267)

Setup clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

Depth at front Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999

Countersinking offset at front Q359 (incremental): Distance by which the TNC moves the tool center away from the stud center. Input range 0 to 99999.9999

Workpiece surface coordinate Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Feed rate for countersinking Q254: Traversing speed of the tool during countersinking in mm/min. Input range: 0 to 99999.999, alternatively FAUTO, FU.

Feed rate for milling Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999, alternatively FAUTO.

Example: NC blocks

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

Example: Thread milling

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

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

Program sequence
- Centering
- Drilling
- Tapping

```
0 BEGIN PGM 1 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20          ;Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0          ;Definition of workpiece blank
3 TOOL DEF 1 L+0 R+4
4 TOOL DEF 2 L+0 2.4
5 TOOL DEF 3 L+0 R+3
6 TOOL CALL 1 Z S5000                   ;Tool call of centering drill
7 L Z+10 R0 F5000                       ;Move tool to clearance height (enter a value for F).
8 SEL PATTERN "TAB1"                     ;The TNC positions to the clearance height after every cycle
9 CYCL DEF 200 DRILLING                ;Defining point tables
   Q200=2 ;SETUP CLEARANCE
   Q201=-2 ;DEPTH
   Q206=150 ;FEED RATE FOR PLNGN
   Q202=2 ;PLUNGING DEPTH
   Q210=0 ;Dwell time at top
   Q203=+0 ;SURFACE COORDINATE
0 must be entered here, effective as defined in point table
```
<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>CYCL CALL PAT F5000 M3</td>
<td>Feed rate between points: 5000 mm/min</td>
</tr>
<tr>
<td>11</td>
<td>L Z+100 R0 FMAX M6</td>
<td>Retract the tool, change the tool</td>
</tr>
<tr>
<td>12</td>
<td>TOOL CALL 2 Z S5000</td>
<td>Call tool: drill</td>
</tr>
<tr>
<td>13</td>
<td>L Z+10 R0 F5000</td>
<td>Move tool to clearance height (enter a value for F)</td>
</tr>
<tr>
<td>14</td>
<td>CYCL DEF 200 DRILLING</td>
<td>Cycle definition: drilling</td>
</tr>
<tr>
<td>15</td>
<td>CYCL CALL PAT F5000 M3</td>
<td>Cycle call in connection with point table TAB1.PNT</td>
</tr>
<tr>
<td>16</td>
<td>L Z+100 R0 FMAX M6</td>
<td>Retract the tool, change the tool</td>
</tr>
<tr>
<td>17</td>
<td>TOOL CALL 3 Z S200</td>
<td>Tool call for tap</td>
</tr>
<tr>
<td>18</td>
<td>L Z+50 R0 FMAX</td>
<td>Move tool to clearance height</td>
</tr>
<tr>
<td>19</td>
<td>CYCL DEF 206 TAPPING NEW</td>
<td>Cycle definition for tapping</td>
</tr>
<tr>
<td>20</td>
<td>CYCL CALL PAT F5000 M3</td>
<td>Cycle call in connection with point table TAB1.PNT</td>
</tr>
<tr>
<td>21</td>
<td>L Z+100 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>22</td>
<td>END PGM 1 MM</td>
<td></td>
</tr>
</tbody>
</table>
## Point table TAB1.PNT

<table>
<thead>
<tr>
<th>NRXYZ</th>
<th>0+10+10+0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1+40+30+0</td>
</tr>
<tr>
<td></td>
<td>2+90+10+0</td>
</tr>
<tr>
<td></td>
<td>3+80+30+0</td>
</tr>
<tr>
<td></td>
<td>4+80+65+0</td>
</tr>
<tr>
<td></td>
<td>5+90+90+0</td>
</tr>
<tr>
<td></td>
<td>6+10+90+0</td>
</tr>
<tr>
<td></td>
<td>7+20+55+0</td>
</tr>
</tbody>
</table>

[END]
Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling
5.1 Basics

Overview

The TNC offers 6 cycles for machining pockets, studs and slots:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>251 RECTANGULAR POCKET</td>
<td>251</td>
<td>Page 141</td>
</tr>
<tr>
<td>Roughing/finishing cycle with selection of machining operation and helical plunging</td>
<td></td>
<td></td>
</tr>
<tr>
<td>252 CIRCULAR POCKET</td>
<td>252</td>
<td>Page 146</td>
</tr>
<tr>
<td>Roughing/finishing cycle with selection of machining operation and helical plunging</td>
<td></td>
<td></td>
</tr>
<tr>
<td>253 SLOT MILLING</td>
<td>253</td>
<td>Page 150</td>
</tr>
<tr>
<td>Roughing/finishing cycle with selection of machining operation and reciprocal plunging</td>
<td></td>
<td></td>
</tr>
<tr>
<td>254 CIRCULAR SLOT</td>
<td>254</td>
<td>Page 155</td>
</tr>
<tr>
<td>Roughing/finishing cycle with selection of machining operation and reciprocal plunging</td>
<td></td>
<td></td>
</tr>
<tr>
<td>256 RECTANGULAR STUD</td>
<td>256</td>
<td>Page 160</td>
</tr>
<tr>
<td>Roughing/finishing cycle with stepover, if multiple passes are required</td>
<td></td>
<td></td>
</tr>
<tr>
<td>257 CIRCULAR STUD</td>
<td>257</td>
<td>Page 164</td>
</tr>
<tr>
<td>Roughing/finishing cycle with stepover, if multiple passes are required</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
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 into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with Parameter Q366.

2. The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowances (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 setup 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. Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.

6. Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.
Please note while programming:

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

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

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed CYCL CALL POS X... Y... or in U and V if you programmed CYCL CALL POS U... V...

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd setup 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 setup clearance. Enter the setup clearance so that the tool cannot jam because of chips.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!

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

- **Machining operation (0/1/2)** Q215: Define the machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.

- **First side length** Q218 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999

- **2nd side length** Q219 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999

- **Corner radius** Q220: Radius of the pocket corner. If you have entered 0 here, the TNC assumes that the corner radius is equal to the tool radius. Input range 0 to 99999.9999

- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- **Angle of rotation** Q224 (absolute): Angle by which the entire pocket is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.0000 to 360.0000

- **Pocket position** Q367: Position of the pocket in reference to the position of the tool when the cycle is called:
  - 0: Tool position = Center of pocket
  - 1: Tool position = Lower left corner
  - 2: Tool position = Lower right corner
  - 3: Tool position = Upper right corner
  - 4: Tool position = Upper left corner

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ

- **Climb or up-cut** Q351: Type of milling operation with M3:
  - +1 = climb milling
  - -1 = up-cut milling
  Alternatively PREDEF
5.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251)

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range: -99999.9999 to 99999.9999

- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

- **Finishing allowance for floor** Q369 (incremental): Finishing allowance in the tool axis. Input range 0 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.9999; alternatively **FAUTO, FU, FZ**.

- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively **PREDEF**
Path overlap factor Q370: Q370 x tool radius = stepover factor $k$. Input range: 0.1 to 1.414 alternatively PREDEF.

Plunging strategy Q366: Type of plunging strategy:

0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle $\text{ANGLE}$ defined in the tool table.

1 = helical plunging. In the tool table, the plunging angle $\text{ANGLE}$ for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.

2 = reciprocating plunge. In the tool table, the plunging angle $\text{ANGLE}$ 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.

Alternative: PREDEF

Feed rate for finishing Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

Example: NC blocks

<table>
<thead>
<tr>
<th>8 CYCL DEF 251 RECTANGULAR POCKET</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q218=80 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q219=60 ;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q220=5 ;CORNER RADIUS</td>
</tr>
<tr>
<td>Q368=0.2 ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q224=+0 ;ANGLE OF ROTATION</td>
</tr>
<tr>
<td>Q367=0 ;POCKET POSITION</td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q351=+1 ;CLimb OR UP-CUT</td>
</tr>
<tr>
<td>Q201=-20 ;DEPTH</td>
</tr>
<tr>
<td>Q202=5 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q369=0.5 ;ALLOWANCE FOR FLOOR</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLUNGING</td>
</tr>
<tr>
<td>Q338=5 ;INFEED FOR FINISHING</td>
</tr>
<tr>
<td>Q200=2 ;SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+0 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q370=1 ;TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q366=1 ;PLUNGE</td>
</tr>
<tr>
<td>Q385=500 ;FEED RATE FOR FINISHING</td>
</tr>
</tbody>
</table>

9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
5.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)

Cycle run

Use Cycle 252 CIRCULAR POCKET to completely 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 tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with Parameter Q366.
2. The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowances (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 setup 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. Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.
6. Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.
Please note while programming:

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

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

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed CYCL CALL POS X... Y... or in U and V if you programmed CYCL CALL POS U... V...

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd setup 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 setup clearance. Enter the setup clearance so that the tool cannot jam because of chips.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.
5.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)

Cycle parameters

- **Machining operation (0/1/2)** Q215: Define the machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.

- **Circle diameter** Q223: Diameter of the finished pocket. Input range 0 to 99999.9999

- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999; alternatively **FAUTO, FU, FZ**

- **Climb or up-cut** Q351: Type of milling operation with M3:
  - +1 = climb milling
  - −1 = up-cut milling
  Alternatively **PREDEF**

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range: -99999.9999 to 99999.9999

- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

- **Finishing allowance for floor** Q369 (incremental): Finishing allowance in the tool axis. Input range 0 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.999; alternatively **FAUTO, FU, FZ**.

- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
Setup clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF.

Workpiece surface coordinate Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999.

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF.

Path overlap factor Q370: $Q370 \times \text{tool radius} = \text{stepover factor } k$. Input range: 0.1 to 1.414, alternatively PREDEF.

Plunging strategy Q366: Type of plunging strategy:
- 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle $\text{ANGLE}$ defined in the tool table.
- 1 = helical plunging. In the tool table, the plunging angle $\text{ANGLE}$ for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
- Alternative: PREDEF

Feed rate for finishing Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

Example: NC blocks

```
8 CYCL DEF 252 CIRCULAR POCKET
Q215=0 ;MACHINING OPERATION
Q223=60 ;CIRCLE DIAMETER
Q368=0.2 ;ALLOWANCE FOR SIDE
Q207=500 ;FEED RATE FOR MILLING
Q351=+1 ;CLIMB OR UP-CUT
Q201=-20 ;DEPTH
Q202=5 ;PLUNGING DEPTH
Q369=0.1 ;ALLOWANCE FOR FLOOR
Q206=150 ;FEED RATE FOR PLUNGING
Q338=5 ;INFEED FOR FINISHING
Q200=2 ;SETUP CLEARANCE
Q203=+0 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q370=1 ;TOOL PATH OVERLAP
Q366=1 ;PLUNGE
Q385=500 ;FEED RATE FOR FINISHING
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```
5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253)

Cycle run

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

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

Roughing

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

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

3. This process is repeated until the slot depth is reached.

Finishing

4. 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 right slot arc.

5. Then the TNC finishes the floor of the slot from the inside out. The slot 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 (slot position).

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed CYCL CALL POS X... Y... or in U and V if you programmed CYCL CALL POS U... V...

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd setup 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 setup 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 inside out. You can therefore mill any slots with small tools, too.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup 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!
5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253)

Cycle parameters

- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.

- **Slot length** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range 0 to 99999.9999

- **Slot width** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999

- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane.

- **Angle of rotation** Q374 (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.000 to 360.000

- **Slot position (0/1/2/3/4)** Q367: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position = Center of slot
  1: Tool position = Left end of slot
  2: Tool position = Center of left slot circle
  3: Tool position = Center of right slot circle
  4: Tool position = Right end of slot

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ

- **Climb or up-cut** Q351: Type of milling operation with M3:
  +1 = climb milling
  -1 = up-cut milling
  Alternatively PREDEF
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999

- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

- **Finishing allowance for floor** Q369 (incremental): Finishing allowance in the tool axis. Input range 0 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ.

- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
Setup clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

Workpiece surface coordinate Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Plunging strategy Q366: Type of plunging strategy:

- 0 = vertical plunging. The 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. Plunge on a helical path only if there is enough space.
- 2 = reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
- Alternative: PREDEF

Feed rate for finishing Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

Example: NC blocks

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

9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
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 (parameters Q368 and Q369) into account.
3. This process is repeated until the slot depth is reached.

Finishing

4. 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.
5. Then the TNC finishes the floor of the slot from the inside out. The slot 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 with radius compensation R0. Define Parameter Q367 (Reference for slot position) appropriately.

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed CYCL CALL POS X... Y... or in U and V if you programmed CYCL CALL POS U... V...

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd setup clearance).

At the end of the cycle the TNC merely moves the tool in working plane back to the center of the pitch circle; 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 setup 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.

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 setup 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 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.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!

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

- **Machining operation (0/1/2) Q215:** Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.

- **Slot width Q219** (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999

- **Finishing allowance for side Q368** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- **Pitch circle diameter Q375:** Enter the diameter of the pitch circle. Input range 0 to 99999.9999

- **Reference for slot position (0/1/2/3) Q367:** Position of the slot in reference to the position of the tool when the cycle is called:
  0: The 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 circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
  2: Tool position = Center of center line. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
  3: Tool position = Center of right slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.

- **Center in 1st axis Q216** (absolute): Center of the pitch circle in the reference axis of the working plane. **Only effective if Q367 = 0.** Input range: -99999.9999 to 99999.9999

- **Center in 2nd axis Q217** (absolute): Center of the pitch circle in the minor axis of the working plane. **Only effective if Q367 = 0.** Input range: -99999.9999 to 99999.9999

- **Starting angle Q376** (absolute): Enter the polar angle of the starting point. Input range -360.000 to 360.000

- **Angular length Q248** (incremental): Enter the angular length of the slot. Input range 0 to 360.000
5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)

- **Stepping angle** Q378 (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. Input range -360.000 to 360.000

- **Number of repetitions** Q377: Number of machining operations on a pitch circle. Input range 1 to 99999

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ

- **Climb or up-cut** Q351: Type of milling operation with M3:
  - +1 = climb milling
  - -1 = up-cut milling
  Alternatively PREDEF

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999

- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

- **Finishing allowance for floor** Q369 (incremental): Finishing allowance in the tool axis. Input range 0 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ.

- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
Setup clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

Workpiece surface coordinate Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Plunging strategy Q366: Type of plunging strategy:

0 = vertical plunging. The 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. Plunge on a helical path only if there is enough space.

2 = reciprocating plunge. 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 TNC can only plunge reciprocally once the traversing length on the circular arc is at least three times the tool diameter.

Alternative: PREDEF

Feed rate for finishing Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ.

Example: NC blocks

```
8 CYCL DEF 254 CIRCULAR SLOT
Q215=0 ;MACHINING OPERATION
Q219=12 ;SLOT WIDTH
Q368=0.2 ;ALLOWANCE FOR SIDE
Q375=80 ;PITCH CIRCLE DIA.
Q367=0 ;REF. SLOT POSITION
Q216=+50 ;CENTER IN 1ST AXIS
Q217=+50 ;CENTER IN 2ND AXIS
Q376=+45 ;STARTING ANGLE
Q378=0 ;STEPPING ANGLE
Q377=1 ;NUMBER OF OPERATIONS
Q207=500 ;FEED RATE FOR MILLING
Q351=+1 ;CLimb OR UP-CUT
Q201=-20 ;DEPTH
Q202=5 ;PLUNGING DEPTH
Q369=0.1 ;ALLOWANCE FOR FLOOR
Q206=150 ;FEED RATE FOR PLUNGING
Q338=5 ;INFEED FOR FINISHING
Q200=2 ;SETUP CLEARANCE
Q203=+0 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q366=1 ;PLUNGE
Q385=500 ;FEED RATE FOR FINISHING
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```
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) in the positive X direction to the starting position for the stud machining. The starting position is 2 mm to the right of the unmachined stud.

2. If the tool is at the 2nd setup clearance, it moves at rapid traverse \( F_{MAX} \) to the setup clearance, and from there it 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.

5. If further stepovers are required the tool then departs the contour on a tangential path and returns to the starting point of stud machining.

6. The TNC then plunges the tool to the next plunging depth, and machines the stud at this depth.

7. This process is repeated until the programmed stud depth is reached.

8. At the end of the cycle, the TNC merely positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position.
Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation \( R0 \). Note Parameter Q367 (stud position).

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd setup 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, the TNC positions the tool back to the setup clearance, or to the 2nd setup clearance if one was programmed.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!

Leave enough room next to the stud for the approach motion. Minimum: tool diameter + 2 mm
Cycle parameters

- **First side length** Q218: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999

- **Workpiece blank side length 1** Q424: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length**. The TNC performs multiple stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap Q370). The TNC always calculates a constant stepover. Input range 0 to 99999.9999

- **Second side length** Q219: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **Second side length**. The TNC performs multiple stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap Q370). The TNC always calculates a constant stepover. Input range 0 to 99999.9999

- **Workpiece blank side length 2** Q425: Length of the stud blank, parallel to the minor axis of the working plane. Input range 0 to 99999.9999

- **Corner radius** Q220: Radius of the stud corner. Input range 0 to 99999.9999

- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane, is left over after machining. Input range 0 to 99999.9999

- **Angle of rotation** Q224 (absolute): Angle by which the entire stud 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

- **Stud position** Q367: Position of the stud in reference to the position of the tool when the cycle is called:
  0: Tool position = Center of stud
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner
**Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ.

**Climb or up-cut** Q351: Type of milling operation with M3:
+1 = climb milling
–1 = up-cut milling
Alternatively PREDEF

**Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999

**Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

**Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.9999; alternatively FMAX, FAUTO, FU, FZ.

**Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

**Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

**2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

**Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414 alternatively PREDEF.

---

**Example: NC blocks**

<table>
<thead>
<tr>
<th>Block</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CYCL DEF 256 RECTANGULAR STUD</td>
<td></td>
</tr>
<tr>
<td>Q218=60</td>
<td>;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q424=74</td>
<td>;WORKPC. BLANK SIDE 1</td>
</tr>
<tr>
<td>Q219=40</td>
<td>;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q425=60</td>
<td>;WORKPC. BLANK SIDE 2</td>
</tr>
<tr>
<td>Q220=5</td>
<td>;CORNER RADIUS</td>
</tr>
<tr>
<td>Q368=0.2</td>
<td>;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q224=+0</td>
<td>;ANGLE OF ROTATION</td>
</tr>
<tr>
<td>Q367=0</td>
<td>;STUD POSITION</td>
</tr>
<tr>
<td>Q207=500</td>
<td>;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q351=+1</td>
<td>;CLIMB OR UP-CUT</td>
</tr>
<tr>
<td>Q201=−20</td>
<td>;DEPTH</td>
</tr>
<tr>
<td>Q202=5</td>
<td>;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PLUNGING</td>
</tr>
<tr>
<td>Q200=2</td>
<td>;SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q370=1</td>
<td>;TOOL PATH OVERLAP</td>
</tr>
</tbody>
</table>

9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
5.7 CIRCULAR STUD (Cycle 257, DIN/ISO: G257)

Cycle run

Use Cycle 257 to machine a circular stud. If a diameter of the workpiece blank is greater than the maximum possible stepover, then the TNC performs multiple stepovers until the finished diameter has been machined.

1. The tool moves from the cycle starting position (stud center) in the positive X direction to the starting position for the stud machining. The starting position is 2 mm to the right of the unmachined stud.

2. If the tool is at the 2nd setup clearance, it moves at rapid traverse \( F_{\text{MAX}} \) to the setup clearance, and from there it 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 diameter 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 diameter, the finished diameter, and the permitted stepover into account. This process is repeated until the defined finished diameter has been reached.

5. If further stepovers are required the tool then departs the contour on a tangential path and returns to the starting point of stud machining.

6. The TNC then plunges the tool to the next plunging depth, and machines the stud at this depth.

7. This process is repeated until the programmed stud depth is reached.

8. At the end of the cycle after tangential departure, 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 (stud center) with radius compensation R0.
- The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd setup 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, the TNC positions the tool back to the setup clearance, or to the 2nd setup clearance if one was programmed.

Danger of collision!

- Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) 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 setup clearance below the workpiece surface!
- Leave enough room next to the stud for the approach motion. Minimum: tool diameter + 2 mm
Cycle parameters

- **Finished part diameter** Q223: Diameter of the completely machined stud. Input range 0 to 99999.9999

- **Workpiece blank diameter** Q222: Diameter of the workpiece blank. Enter the workpiece blank diameter greater than the finished diameter. The TNC performs multiple stepovers if the difference between the workpiece blank diameter and finished diameter is greater than the permitted stepover (tool radius multiplied by path overlap Q370). The TNC always calculates a constant stepover. Input range 0 to 99999.9999

- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ

- **Climb or up-cut** Q351: Type of milling operation with M3:
  
  +1 = climb milling  
  –1 = up-cut milling  

  Alternatively PREDEF
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999

- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.9999; alternatively FMAX, FAUTO, FU, FZ.

- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414 alternatively PREDEF.

---

**Example: 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 MILLING
Q351=+1 ;CLIMB OR UP-CUT
Q201=-20 ;DEPTH
Q202=5 ;PLUNGING DEPTH
Q206=150 ;FEED RATE FOR PLUNGING
Q200=2 ;SETUP CLEARANCE
Q203=+0 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q370=1 ;TOOL PATH OVERLAP
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```
5.8 Programming Examples

Example: Milling pockets, studs and slots

0 BEGIN 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 DEF 1 L+0 R+6
  Define the tool for roughing/finishing
4 TOOL DEF 2 L+0 R+3
  Define slotting mill
5 TOOL CALL 1 Z S3500
  Call the tool for roughing/finishing
6 L Z+250 R0 FMAX
  Retract the tool
### 5.8 Programming Examples

#### Example 1: Rectangular Stud Milling

<table>
<thead>
<tr>
<th>CYCL</th>
<th>DEF</th>
<th>RECTANGULAR STUD</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>CYCL</td>
<td>Define cycle for machining the contour outside</td>
</tr>
<tr>
<td></td>
<td>CALL</td>
<td>X+50  Y+50  Z+0 M3</td>
</tr>
<tr>
<td></td>
<td>POS</td>
<td>Call cycle for machining the contour outside</td>
</tr>
<tr>
<td></td>
<td></td>
<td>M6</td>
</tr>
</tbody>
</table>

#### Example 2: Circular Pocket Milling

<table>
<thead>
<tr>
<th>CYCL</th>
<th>DEF</th>
<th>CIRCULAR POCKET</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>CYCL</td>
<td>Define CIRCULAR POCKET MILLING cycle</td>
</tr>
<tr>
<td></td>
<td>CALL</td>
<td>X+50  Y+50  Z+0 M3</td>
</tr>
<tr>
<td></td>
<td>POS</td>
<td>Call CIRCULAR POCKET MILLING cycle</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FMAX</td>
</tr>
</tbody>
</table>

### Parameters:

- 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 ; ROTATIONAL POSITION
- Q367 = 0 ; STUD POSITION
- Q207 = 250 ; FEED RATE FOR MILLING
- Q351 = +1 ; CLIMB OR UP-CUT
- Q201 = -30 ; DEPTH
- Q202 = 5 ; PLUNGING DEPTH
- Q206 = 250 ; FEED RATE FOR PLUNGING
- Q200 = 2 ; SETUP CLEARANCE
- Q203 = +0 ; SURFACE COORDINATE
- Q204 = 20 ; 2ND SETUP CLEARANCE
- Q370 = 1 ; TOOL PATH OVERLAP
- Q366 = 1 ; PLUNGE
- Q385 = 750 ; FEED RATE FOR FINISHING
Call slotting mill

Define SLOT cycle

MACHINING OPERATION
SLOT WIDTH
ALLOWANCE FOR SIDE
PITCH CIRCLE DIA.
REF. SLOT POSITION
No pre-positioning in X/Y required
CENTER IN 1ST AXIS
CENTER IN 2ND AXIS
STARTING ANGLE
ANGULAR LENGTH
STEPPING ANGLE
Starting point for 2nd slot
NUMBER OF OPERATIONS
FEED RATE FOR MILLING
CLIMB OR UP-CUT
DEPTH
PLUNGING DEPTH
ALLOWANCE FOR FLOOR
FEED RATE FOR PLUNGING
INFEED FOR FINISHING
SETUP CLEARANCE
SURFACE COORDINATE
2ND SETUP CLEARANCE
PLUNGE
Call SLOT cycle
Retract in the tool axis, end program
End program
Fixed Cycles: Pattern Definitions
6.1 Fundamentals

Overview

The TNC provides two cycles for machining point patterns directly:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>220 CIRCULAR PATTERN</td>
<td></td>
<td>Page 173</td>
</tr>
<tr>
<td>221 LINEAR PATTERN</td>
<td></td>
<td>Page 176</td>
</tr>
</tbody>
</table>

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” on page 66) to develop point tables.

More regular point patterns are available with the PATTERN DEF function (see “Pattern Definition PATTERN DEF” on page 58).

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 CIRCULAR PATTERN (Cycle 220, DIN/ISO: G220)

Cycle run

1. The TNC moves the tool at rapid traverse 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 setup clearance above the workpiece surface (spindle axis).

2. From this position the TNC executes the last defined fixed cycle.

3. The tool then approaches on a straight line or circular arc the starting point for the next machining operation. The tool stops at the set-up clearance (or the 2nd setup 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 setup clearance, workpiece surface and 2nd setup clearance that you defined in Cycle 220 will be effective for the selected fixed cycle.
Cycle parameters

- **Center in 1st axis** Q216 (absolute): Center of the pitch circle in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q217 (absolute): Center of the pitch circle in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Pitch circle diameter** Q244: Diameter of the pitch circle. Input range 0 to 99999.9999

- **Starting angle** Q245 (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range -360.000 to 360.000

- **Stopping angle** Q246 (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to full circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range -360.000 to 360.000

- **Stepping angle** Q247 (incremental): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (– = clockwise). Input range -360.000 to 360.000

- **Number of repetitions** Q241: Number of machining operations on a pitch circle. Input range 1 to 99999
Setup clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

Workpiece surface coordinate Q203 (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Moving to clearance height Q301: Definition of how the tool is to move between machining processes.
   0: Move to the setup clearance between operations.
   1: Move to the 2nd setup clearance between machining operations.
   Alternatively PREDEF

Type of traverse? Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between machining operations.
   0: Move between operations on a straight line
   1: Move between operations on the pitch circle

Example: NC blocks

```
53 CYCLE DEF 220 POLAR PATTERN
Q216=+50 ;CENTER IN 1ST AXIS
Q217=+50 ;CENTER 2ND AXIS
Q244=80 ;PITCH CIRCLE DIA.
Q245=+0 ;STARTING ANGLE
Q246=+360 ;STOPPING ANGLE
Q247=+0 ;STEPPING ANGLE
Q241=8 ;NUMBER OF OPERATIONS
Q200=2 ;SETUP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q301=1 ;MOVE TO CLEARANCE
Q365=0 ;TYPE OF TRAVERSE
```
6.3 LINEAR PATTERN (Cycle 221, DIN/ISO: G221)

Cycle run

1. The TNC automatically moves the tool from its current position to the starting point for the first machining operation.
   Sequence:
   - Move to the 2nd set-up clearance (spindle axis)
   - Approach the starting point in the spindle axis.
   - Move to the setup 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 the setup clearance (or the 2nd setup 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 setup clearance, workpiece surface, 2nd setup clearance and the rotational position that you defined in Cycle221 will be effective for the selected fixed cycle.

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

- **Starting point 1st axis** Q225 (absolute): Coordinate of the starting point in the reference axis of the working plane.
- **Starting point 2nd axis** Q226 (absolute): Coordinate of the starting point in the minor axis of the working plane.
- **Spacing in 1st axis** Q237 (incremental): Spacing between each point on a line.
- **Spacing in 2nd axis** Q238 (incremental): Spacing between each line.
- **Number of columns** Q242: Number of machining operations on a line.
- **Number of lines** Q243: Number of passes.
- **Rotational position** Q224 (absolute): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point.
- **Setup clearance** Q200 (incremental): Distance between tool tip and workpiece surface, alternatively PREDEF.
- **Workpiece surface coordinate** Q203 (absolute): Coordinate of the workpiece surface.
- **2nd setup clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur, alternatively PREDEF.
- **Moving to clearance height** Q301: Definition of how the tool is to move between machining processes.
  - 0: Move to the setup clearance between operations.
  - 1: Move to the 2nd setup clearance between machining operations.
  - Alternatively PREDEF

**Example: NC blocks**

```
54 CYCL DEF 221 CARTESIAN PATTERN
Q225=+15 ;STARTING PNT 1ST AXIS
Q226=+15 ;STARTING 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 ;ROTATIONAL POSITION
Q200=2 ;SETUP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SETUP CLEARANCE
Q301=1 ;MOVE TO CLEARANCE
```
### 6.4 Programming Examples

#### Example: Circular hole patterns

```plaintext
0 BEGIN PGM PATTERN MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
   Definition of workpiece blank
2 BLK FORM 0.2 Y+100 Y+100 Z+0
3 TOOL DEF 1 L+0 R+3
   Tool definition
4 TOOL CALL 1 Z S3500
   Tool call
5 L Z+250 R0 FMAX M3
   Retract the tool
6 CYCL DEF 200 DRILLING
   Cycle definition: drilling
   Q200=2 ;SETUP CLEARANCE
   Q201=-15 ;DEPTH
   Q206=250 ;FEED RATE FOR PLNGN
   Q202=4 ;PLUNGING DEPTH
   Q210=0 ;DWELL TIME
   Q203=+0 ;SURFACE COORDINATE
   Q204=0 ;2ND SET-UP CLEARANCE
   Q211=0.25 ;DWELL TIME AT DEPTH
```
### 7 CYCLE DEF 220 POLAR PATTERN
Define cycle for circular pattern 1, CYCL 200 is called automatically.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q216</td>
<td>+30</td>
<td>CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q217</td>
<td>+70</td>
<td>CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q244</td>
<td>50</td>
<td>PITCH CIRCLE DIA.</td>
</tr>
<tr>
<td>Q245</td>
<td>0</td>
<td>STARTING ANGLE</td>
</tr>
<tr>
<td>Q246</td>
<td>+360</td>
<td>STOPPING ANGLE</td>
</tr>
<tr>
<td>Q247</td>
<td>0</td>
<td>STEPPING ANGLE</td>
</tr>
<tr>
<td>Q241</td>
<td>10</td>
<td>QUANTITY</td>
</tr>
<tr>
<td>Q200</td>
<td>2</td>
<td>SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q203</td>
<td>0</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204</td>
<td>100</td>
<td>2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q301</td>
<td>1</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q365</td>
<td>0</td>
<td>TYPE OF TRAVERSE</td>
</tr>
</tbody>
</table>

### 8 CYCLE DEF 220 POLAR PATTERN
Define cycle for circular pattern 2, CYCL 200 is called automatically.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q216</td>
<td>+90</td>
<td>CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q217</td>
<td>+25</td>
<td>CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q244</td>
<td>70</td>
<td>PITCH CIRCLE DIA.</td>
</tr>
<tr>
<td>Q245</td>
<td>90</td>
<td>STARTING ANGLE</td>
</tr>
<tr>
<td>Q246</td>
<td>+360</td>
<td>STOPPING ANGLE</td>
</tr>
<tr>
<td>Q247</td>
<td>30</td>
<td>STEPPING ANGLE</td>
</tr>
<tr>
<td>Q241</td>
<td>5</td>
<td>QUANTITY</td>
</tr>
<tr>
<td>Q200</td>
<td>2</td>
<td>SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q203</td>
<td>0</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204</td>
<td>100</td>
<td>2ND SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q301</td>
<td>1</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q365</td>
<td>0</td>
<td>TYPE OF TRAVERSE</td>
</tr>
</tbody>
</table>

9 L Z+250 R0 FMAX M2  
Retract in the tool axis, end program

10 END PGM PATTERN MM
6.4 Programming Examples

Fixed Cycles: Pattern Definitions
Fixed Cycles: Contour Pocket, Contour Trains
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 (all contour subprograms) is limited. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. You can program up to 8192 contour elements.

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.

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 ignores feed rates F and miscellaneous functions M.
- 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.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted in useful combinations. Always define both axes of the machining plane in the first block.
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms.

Example: 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 CYCLE DEF 23 FLOOR FINISHING ...
23 CYCL CALL ...
...
26 CYCL DEF 24 SIDE FINISHING ...
27 CYCL CALL ...
...
50 L Z+250 R0 FMAX M2
51 LBL 1 ...
...
55 LBL 0
56 LBL 2 ...
...
60 LBL 0 ...
...
99 END PGM SL2 MM
```
Characteristics of the fixed cycles

- The TNC automatically positions the tool to the setup clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- In order to avoid leaving dwell marks, the TNC inserts a globally definable rounding radius at non-tangential inside corners. The rounding radius, which is entered in Cycle 20, affects the tool center point path, meaning that it would increase a rounding defined by the tool radius (applies to rough-out and side finishing).
- 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 tool 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 miscellaneous functions M109 and M110 (feed rate at circular arcs) are not effective within SL cycles, even if you programmed these before a cycle call.

With bit 4 in MP7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

- **Bit 4 = 0:**
  - At the end of the cycle, the TNC at first positions the tool in the tool axis at the clearance height (Q7) defined in the cycle, and then to the position in the working plane at which the tool was located when the cycle was called.
- **Bit 4 = 1:**
  - At the end of the cycle, the TNC always positions the tool in the tool axis at the clearance height (Q7) defined in the cycle. Ensure that no collisions can occur during the following positioning movements!

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle 20.
## Overview

### Enhanced cycles:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 CONTOUR TRAIN</td>
<td>25</td>
<td>201</td>
</tr>
<tr>
<td>270 CONTOUR TRAIN DATA</td>
<td>270</td>
<td>203</td>
</tr>
<tr>
<td>275 TROCHOIDAL SLOT</td>
<td>275</td>
<td>205</td>
</tr>
</tbody>
</table>
7.2 CONTOUR GEOMETRY (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.

Before programming, note the following:

- 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 254.
7.3 Overlapping Contours

Basics

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.

Example: NC blocks

```
12 CYCL DEF 14.0 CONTOUR GEOMETRY
13 CYCL DEF 14.1 CONTOUR LABEL1/2/3/4
```
7.3 Overlapping Contours

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 $S_1$ and $S_2$. They do not have to be programmed.

The pockets are programmed as full circles.

**Subprogram 1: Pocket A**

```plaintext
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
```

```plaintext
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**

```plaintext
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
```

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

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

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

**Surface A:**

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

**Surface B:**

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

Surface A is to be machined without the portion overlapped by B:
- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.

**Surface A:**

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

**Surface B:**

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

Area of intersection

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

**Surface A:**

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

**Surface B:**

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```
7.4 CONTOUR DATA (Cycle 20, DIN/ISO: G120)

Please note while programming:

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

Cycle 20 is DEF active, which means that it becomes effective as soon as it is defined in the part program.

The 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.

The machining data entered in Cycle 20 are valid for Cycles 21 to 24.

If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q20 cannot be used as program parameters.
Cycle parameters

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

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

---

**Example: 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 ; SETUP CLEARANCE
Q7=+80 ; CLEARANCE HEIGHT
Q8=0.5 ; ROUNDS RADIUS
Q9=+1 ; DIRECTION
```
7.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121)

**Cycle run**

1. The tool drills from the current position to the first plunging depth at the programmed feed rate $F$.

2. Then the tool retracts at rapid traverse $F_{MAX}$ to the starting position and advances again to the first plunging depth minus the advanced stop distance $t$.

3. The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: $t = 0.6$ mm
   - At a total hole depth exceeding 30 mm: $t = \text{hole depth} / 50$
   - Maximum advanced stop distance: 7 mm

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

5. The TNC repeats this process (1 to 4) until the programmed depth is reached.

6. After a dwell time at the hole bottom, the tool is returned to the starting position at rapid traverse $F_{MAX}$ for chip breaking.

**Application**

Cycle 21 is for PILOT DRILLING of the cutter infeed points. It accounts for the allowance for side and the allowance for floor as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for roughing.

**Please note while programming:**

Before programming, note the following:

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.
Cycle parameters

- **Plunging depth Q10** (incremental): Dimension by which the tool drills in each infeed (negative sign for negative working direction). Input range: -99999.9999 to 99999.9999.

- **Feed rate for plunging Q11**: Drilling feed rate in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

- **Rough-out tool number/name Q13 or QS13**: Number or name of rough-out tool. Input range 0 to 32767.9 if a number is entered; maximum 16 characters if a name is entered.

Example: NC blocks

```
58 CYCL DEF 21 PILOT DRILLING
Q10=+5 ; PLUNGING DEPTH
Q11=100 ; FEED RATE FOR PLUNGING
Q13=1 ; ROUGH-OUT TOOL
```
7.6 ROUGH-OUT (Cycle 22, DIN/ISO: G122)

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 the inside outward at the milling feed rate Q12.
3. The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B).
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 TNC retracts the tool to the clearance height.
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 geometry), 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.

Feed rate reduction through parameter Q401 is an FCL3 function and is not automatically available after a software update (see “Feature content level (upgrade functions)” on page 8).
7.6 ROUGH-OUT (Cycle 22, DIN/ISO: G122)

**Cycle parameters**

- **Plunging depth Q10** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999.
- **Feed rate for plunging Q11**: Plunging feed rate in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.
- **Feed rate for roughing Q12**: Milling feed rate in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.
- **Coarse roughing tool Q18 or QS18**: Number or name of the tool with which the TNC has already coarse-roughed the contour. Switch to name input: Press the TOOL NAME soft key. The TNC automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter “0”; if you enter a number or a name, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut; For this purpose you must enter the tool length LCUTS in the tool table TOOL.T and define the maximum plunging ANGLE of the tool. The TNC will otherwise generate an error message. Input range 0 to 32767.9 if a number is entered; maximum 16 characters if a name is entered.
- **Reciprocation feed rate Q19**: Traversing speed of the tool in mm/min during reciprocating plunge cut. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.
- **Retraction feed rate Q208**: Traversing speed of the tool in mm/min when retracting after machining. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q12. Input range 0 to 99999.9999, alternatively FMAX, FAUTO, PREDEF.

**Example: NC blocks**

```
59 CYCL DEF 22 ROUGH-OUT
Q10=+5 ;PLUNGING DEPTH
Q11=100 ;FEED RATE FOR PLUNGING
Q12=750 ;FEED RATE FOR ROUGHING
Q18=1 ;COARSE ROUGHING TOOL
Q19=150 ;RECIROCATION FEED RATE
Q208=9999 ;RETRACTION FEED RATE
Q401=80 ;FEED RATE REDUCTION
Q404=0 ;FINE ROUGH STRATEGY
```
**Feed rate factor in %**: Q401: Percentage factor by which the TNC reduces the machining feed rate (Q12) as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (Q2) specified in Cycle 20. The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machining time should be reduced in total. Input range 0.0001 to 100.0000

**Fine-roughing strategy** Q404: Specify how the TNC should move the tool during fine roughing when the radius of the fine-roughing tool is larger than half the coarse roughing tool.

- Q404 = 0
  Move the tool along the contour at the current depth between areas that need to be fine-roughed.

- Q404 = 1
  Between areas that need to be fine-roughed, retract the tool to safety clearance and move to the starting point of the next area to be rough-milled.
7.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123)

**Cycle run**

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.

**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.

**Cycle parameters**

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

**Example: NC blocks**

```
60 CYCL DEF 23 FLOOR FINISHING
Q11=100 ;FEED RATE FOR PLNGNG
Q12=350 ;FEED RATE FOR ROUGHING
Q208=99999 ;RETRACTION FEED RATE
```
7.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124)

Cycle run

The individual subcontours are approached and departed on a tangential arc. The TNC finishes each subcontour separately.

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.

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 TNC executes the positioning logic to the starting point of the finishing operation as follows: approach the starting point in the working plane, then move to depth in tool axis direction.

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.
Cycle parameters

- **Direction of rotation? Clockwise = –1 Q9:**
  - Machining direction:
    - +1: Counterclockwise
    - –1: Clockwise
  - Alternatively **PREDEF**

- **Plunging depth** Q10 (incremental): Infeed per cut.
  - Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging** Q11: Traversing speed of the tool during plunging.
  - Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**

- **Feed rate for roughing** Q12: Milling feed rate.
  - Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**

- **Finishing allowance for side** Q14 (incremental): Enter the allowed material for several finish-milling operations.
  - If you enter Q14 = 0, the remaining finishing allowance will be cleared.
  - Input range -99999.9999 to 99999.9999

**Example: NC blocks**

```plaintext
61 CYCLE DEF 24 SIDE FINISHING
Q9=+1 ; DIRECTION
Q10=+5 ; PLUNGING DEPTH
Q11=100 ; FEED RATE FOR PLNGNG
Q12=350 ; FEED RATE FOR ROUGHING
Q14=+0 ; ALLOWANCE FOR SIDE
```
7.9 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125)

Cycle run

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

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

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

Please note while programming:

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

When you use Cycle 25 CONTOUR TRAIN you can define only one contour program in Cycle 14 CONTOUR.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements in one SL cycle.

The TNC does not need Cycle 20 CONTOUR DATA in conjunction with Cycle 25.

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

Danger of collision!

To avoid collisions,

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

- **Milling depth** $Q_1$ (incremental): Distance between workpiece surface and contour floor. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** $Q_3$ (incremental): Finishing allowance in the working plane. Input range -99999.9999 to 99999.9999
- **Workpiece surface coordinate** $Q_5$ (absolute): Absolute coordinate of the workpiece surface referenced to the workpiece datum. Input range -99999.9999 to 99999.9999
- **Clearance height** $Q_7$ (absolute): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle. Input range -99999.9999 to 99999.9999, alternatively PREDEF
- **Plunging depth** $Q_{10}$ (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** $Q_{11}$: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ
- **Feed rate for milling** $Q_{12}$: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ
- **Climb or up-cut?** Up-cut = $-1$ $Q_{15}$: Climb milling: Input value = +1
  Up-cut milling: Input value = $-1$
  To enable climb milling and up-cut milling alternately in several infeeds: Input value = 0

---

**Example: NC blocks**

```
G2 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 PLUNGING
Q12=350 ; FEED RATE FOR MILLING
Q15=-1 ; CLIMB OR UP-CUT
```
7.10 CONTOUR DATA (Cycle 270, DIN/ISO: G270)

Please note while programming:

If desired, you can use this cycle to specify various properties of Cycle 25, CONTOUR TRAIN.

**Before programming, note the following:**

Cycle 270 is DEF active, which means that it becomes effective as soon as it is defined in the part program.

The TNC resets Cycle 270 as soon as you define another SL cycle (with the exception of Cycle 25).

If Cycle 270 is used, do not define any radius compensation in the contour subprogram.

Approach and departure properties are always performed identically (symmetrically) by the TNC.

Define Cycle 270 before Cycle 25.
**Cycle parameters**

- **Type of approach/departure** Q390: 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.

- **Radius compensation (0=R0/1=RL/2=RR)** Q391: Definition of the radius compensation:
  - Q391 = 0: Machine the defined contour without radius compensation
  - Q391 = 1: Machine the defined contour with compensation to the left
  - Q391 = 2: Machine the defined contour with compensation to the right

- **Approach/departure radius** Q392: Only in effect if tangential approach on a circular path was selected. Radius of the approach/departure arc. Input range 0 to 99999.9999

- **Center angle** Q393: Only in effect if tangential approach on a circular path was selected. Angular length of the approach arc. Input range 0 to 99999.9999

- **Distance to auxiliary point** Q394: Only in effect if tangential approach on a straight line or right-angle approach was selected. Distance to the auxiliary point from which the TNC is to approach the contour. Input range 0 to 99999.9999

**Example: 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

Example: TROCHOIDAL SLOT scheme

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

**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 closed slots**
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 you use Cycle 275 TROCHOIDAL SLOT, you can define only one contour program 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 8192 contour elements in one SL cycle.

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

The miscellaneous functions M109 and M110 are not effective when machining a contour with Cycle 275.

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

Cycle parameters

- **Machining operation (0/1/2)** Q215: Define the machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing
  The TNC also executes side finishing if the finishing allowance (Q368) defined is 0.

- **Slot width** Q219: Enter the slot width; If you enter a slot width that equals the tool diameter, the TNC will only machine the contour outline. Input range 0 to 99999.999

- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane.

- **Infeed per rev.** Q436 absolute: Value by which the TNC moves the tool in the machining direction per revolution. Input range: 0 to 99999.999

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.999; alternatively FAUTO, FU, FZ

- **Climb or up-cut** Q351: Type of milling operation with M3:
  - +1 = climb milling
  - –1 = up-cut milling
  Alternatively PREDEF
7.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999

- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999

- **Feed rate for finishing** Q385: Traversing speed of the tool during side finishing in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.
Setup clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999, alternatively PREDEF

Workpiece surface coordinate Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Plunging strategy Q366: Type of plunging strategy:
- 0 = vertical plunging. The 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.
- Alternative: PREDEF

Example: NC blocks

```
8 CYCL DEF 275 TROCHOIDAL SLOT
  Q215=0 ;MACHINING OPERATION
  Q219=12 ;SLOT WIDTH
  Q368=0.2 ;ALLOWANCE FOR SIDE
  Q436=2 ;INFEED PER REVOLUTION
  Q207=500 ;FEED RATE FOR MILLING
  Q351=+1 ;CLIMB OR UP-CUT
  Q201=-20 ;DEPTH
  Q202=5 ;PLUNGING DEPTH
  Q206=150 ;FEED RATE FOR PLUNGING
  Q338=5 ;INFEED FOR FINISHING
  Q385=500 ;FEED RATE FOR FINISHING
  Q200=2 ;SETUP CLEARANCE
  Q203=+0 ;SURFACE COORDINATE
  Q204=50 ;2ND SETUP CLEARANCE
  Q366=2 ;PLUNGE
9 CYCL CALL FMAX M3
```
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
3 TOOL CALL 1 Z S2500
4 L Z+250 R0 FMAX
5 CYCL DEF 14.0 CONTOUR GEOMETRY
6 CYCL DEF 14.1 CONTOUR LABEL 1
7 CYCL DEF 20 CONTOUR DATA
   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 ;SETUP CLEARANCE
   Q7=+100 ;CLEARANCE HEIGHT
   Q8=0.1 ;ROUNDING RADIUS
   Q9=-1 ;DIRECTION
8 CYCL DEF 22 ROUGH-OUT

Cycle definition: Coarse roughing

Q10=5 ; PLUNGING DEPTH
Q11=100 ; FEED RATE FOR PLUNGING
Q12=350 ; FEED RATE FOR ROUGHING
Q18=0 ; COARSE ROUGHING TOOL
Q19=150 ; RECIPROCATION FEED RATE
Q208=30000 ; RETRACTION FEED RATE
Q401=100 ; FEED RATE FACTOR
Q404=0 ; FINE ROUGH STRATEGY

9 CYCL CALL M3
Cycle call: Coarse roughing

10 L Z+250 R0 FMAX M6
Tool change

11 TOOL CALL 2 Z S3000
Tool call: Fine roughing tool, diameter 15

12 CYCL DEF 22 ROUGH-OUT
Define the fine roughing cycle

Q10=5 ; PLUNGING DEPTH
Q11=100 ; FEED RATE FOR PLUNGING
Q12=350 ; FEED RATE FOR ROUGHING
Q18=1 ; COARSE ROUGHING TOOL
Q19=150 ; RECIPROCATION FEED RATE
Q208=30000 ; RETRACTION FEED RATE
Q401=100 ; FEED RATE FACTOR
Q404=0 ; FINE ROUGH STRATEGY

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
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL CALL 1 Z S2500
4 L Z+250 R0 FMAX
5 CYCL DEF 14.0 CONTOUR GEOMETRY
6 CYCL DEF 14.1 CONTOUR LABEL1/2/3/4
7 CYCL DEF 20 CONTOUR DATA
   Q1=-20 ;MILLING DEPTH
   Q2=1 ;TOOL PATH OVERLAP
   Q3=+0.5 ;ALLOWANCE FOR SIDE
   Q4=+0.5 ;ALLOWANCE FOR FLOOR
   Q5=+0.5 ;ALLOWANCE FOR FLOOR
   Q6=2 ;SURFACE COORDINATE
   Q7=+100 ;SETUP CLEARANCE
   Q8=0.1 ;CLEARANCE HEIGHT
   Q9=-1 ;DIRECTION
<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>CYCL DEF 21 PILOT DRILLING&lt;br&gt;Q10=5 ;PLUNGING DEPTH&lt;br&gt;Q11=250 ;FEED RATE FOR PLNGNG&lt;br&gt;Q13=2 ;ROUGH-OUT TOOL</td>
</tr>
<tr>
<td>9</td>
<td>CYCL CALL M3</td>
</tr>
<tr>
<td>10</td>
<td>L +250 R0 FMAX M6</td>
</tr>
<tr>
<td>11</td>
<td>TOOL CALL 2 Z S3000</td>
</tr>
<tr>
<td>12</td>
<td>CYCL DEF 22 ROUGH-OUT&lt;br&gt;Q10=5 ;PLUNGING DEPTH&lt;br&gt;Q11=100 ;FEED RATE FOR PLNGNG&lt;br&gt;Q12=350 ;FEED RATE FOR ROUGHING&lt;br&gt;Q18=0 ;COARSE ROUGHING TOOL&lt;br&gt;Q19=150 ;RECIPROCATION FEED RATE&lt;br&gt;Q208=30000 ;RETRACTION FEED RATE&lt;br&gt;Q401=100 ;FEED RATE FACTOR&lt;br&gt;Q404=0 ;FINE ROUGH STRATEGY</td>
</tr>
<tr>
<td>13</td>
<td>CYCL CALL M3</td>
</tr>
<tr>
<td>14</td>
<td>CYCL DEF 23 FLOOR FINISHING&lt;br&gt;Q11=100 ;FEED RATE FOR PLNGNG&lt;br&gt;Q12=200 ;FEED RATE FOR ROUGHING&lt;br&gt;Q208=30000 ;RETRACTION FEED RATE</td>
</tr>
<tr>
<td>15</td>
<td>CYCL CALL</td>
</tr>
<tr>
<td>16</td>
<td>CYCLE DEF 24 SIDE FINISHING&lt;br&gt;Q9=+1 ;DIRECTION&lt;br&gt;Q10=5 ;PLUNGING DEPTH&lt;br&gt;Q11=100 ;FEED RATE FOR PLNGNG&lt;br&gt;Q12=400 ;FEED RATE FOR ROUGHING&lt;br&gt;Q14=+0 ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>17</td>
<td>CYCL CALL</td>
</tr>
<tr>
<td>18</td>
<td>L Z+250 R0 FMAX M2</td>
</tr>
<tr>
<td>Line</td>
<td>Code</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>19</td>
<td>LBL 1</td>
</tr>
<tr>
<td>20</td>
<td>CC X+35 Y+50</td>
</tr>
<tr>
<td>21</td>
<td>L X+10 Y+50 RR</td>
</tr>
<tr>
<td>22</td>
<td>C X+10 DR-</td>
</tr>
<tr>
<td>23</td>
<td>LBL 0</td>
</tr>
<tr>
<td>24</td>
<td>LBL 2</td>
</tr>
<tr>
<td>25</td>
<td>CC X+65 Y+50</td>
</tr>
<tr>
<td>26</td>
<td>L X+90 Y+50 RR</td>
</tr>
<tr>
<td>27</td>
<td>C X+90 DR-</td>
</tr>
<tr>
<td>28</td>
<td>LBL 0</td>
</tr>
<tr>
<td>29</td>
<td>LBL 3</td>
</tr>
<tr>
<td>30</td>
<td>L X+27 Y+50 RL</td>
</tr>
<tr>
<td>31</td>
<td>L Y+58</td>
</tr>
<tr>
<td>32</td>
<td>L X+43</td>
</tr>
<tr>
<td>33</td>
<td>L Y+42</td>
</tr>
<tr>
<td>34</td>
<td>L X+27</td>
</tr>
<tr>
<td>35</td>
<td>LBL 0</td>
</tr>
<tr>
<td>36</td>
<td>LBL 4</td>
</tr>
<tr>
<td>39</td>
<td>L X+65 Y+42 RL</td>
</tr>
<tr>
<td>37</td>
<td>L X+57</td>
</tr>
<tr>
<td>38</td>
<td>L X+65 Y+58</td>
</tr>
<tr>
<td>39</td>
<td>L X+73 Y+42</td>
</tr>
<tr>
<td>40</td>
<td>LBL 0</td>
</tr>
<tr>
<td>41</td>
<td>END PGM C21 MM</td>
</tr>
</tbody>
</table>
Example: Contour train

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.</td>
<td>BEGIN PGM C25 MM</td>
<td></td>
</tr>
<tr>
<td>1.</td>
<td>BLK FORM 0.1 Z X+0 Y+0 Z-40</td>
<td>Definition of workpiece blank</td>
</tr>
<tr>
<td>2.</td>
<td>BLK FORM 0.2 X+100 Y+100 Z+0</td>
<td></td>
</tr>
<tr>
<td>3.</td>
<td>TOOL CALL 1 Z S2000</td>
<td>Tool call: Diameter 20</td>
</tr>
<tr>
<td>4.</td>
<td>L Z+250 R0 FMAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>5.</td>
<td>CYCL DEF 14.0 CONTOUR GEOMETRY</td>
<td>Define contour subprogram</td>
</tr>
<tr>
<td>6.</td>
<td>CYCL DEF 14.1 CONTOUR LABEL 1</td>
<td></td>
</tr>
<tr>
<td>7.</td>
<td>CYCL DEF 25 CONTOUR TRAIN</td>
<td>Define machining parameters</td>
</tr>
<tr>
<td>8.</td>
<td>Q1=-20 ;MILLING DEPTH</td>
<td></td>
</tr>
<tr>
<td>9.</td>
<td>Q3=+0 ;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td>10.</td>
<td>Q5=+0 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>11.</td>
<td>Q7=+250 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td>12.</td>
<td>Q10=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>13.</td>
<td>Q11=100 ;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td>14.</td>
<td>Q12=200 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>15.</td>
<td>Q15=+1 ;CLIMB OR UP-CUT</td>
<td></td>
</tr>
<tr>
<td>8.</td>
<td>CYCL CALL M3</td>
<td>Cycle call</td>
</tr>
<tr>
<td>9.</td>
<td>L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
</tbody>
</table>
Contour subprogram

10 LBL 1
11 LBL 0
12 END

7.12 Programming Examples
Fixed Cycles: Cylindrical Surface
### 8.1 Basics

**Overview of cylindrical surface cycles**

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>27 CYLINDER SURFACE</td>
<td></td>
<td>221</td>
</tr>
<tr>
<td>28 CYLINDER SURFACE slot milling</td>
<td></td>
<td>224</td>
</tr>
<tr>
<td>29 CYLINDER SURFACE ridge milling</td>
<td></td>
<td>227</td>
</tr>
<tr>
<td>39 CYLINDER SURFACE outside contour milling</td>
<td></td>
<td>230</td>
</tr>
</tbody>
</table>
8.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, Software Option 1)

Execution of cycle

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.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, is parallel to the Z axis. The path functions L, CHF, CR, RND, APPR (except for APPR LCT) and DEP are available.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type 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 setup clearance and returns to the point of penetration.
4. Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.
5. Then the tool moves to the setup clearance.
Please note while programming:

- The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder. Refer to your machine tool manual.

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The memory capacity for programming an SL cycle is limited. You can program up to 8192 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 (DIN 844).
- The cylinder must be set up centered on the rotary table.
- The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.
- This cycle can also be used in a tilted working plane.
Cycle parameters

- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range: -99999.9999 to 99999.9999

- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Plunging depth** Q10 (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**

- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively **FAUTO, FU, FZ**

- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999

- **Dimension type? ang./lin.** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

---

Example: NC blocks

```
63 CYCL DEF 27 CYLINDER SURFACE
Q1=-8 ;MILLING DEPTH
Q3=+0 ;ALLOWANCE FOR SIDE
Q6=+0 ;SETUP CLEARANCE
Q10=+3 ;PLUNGING DEPTH
Q11=100 ;FEED RATE FOR PLUNGING
Q12=350 ;FEED RATE FOR MILLING
Q16=25 ;RADIUS
Q17=0 ;TYPE OF DIMENSION
```
8.3 CYLINDER SURFACE Slot Milling
(Cycle 28, DIN/ISO: G128, Software Option 1)

Cycle run

This cycle enables you to 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 in parameter Q21 a 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 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.
3 At the end of the contour, the TNC moves the tool to the opposite wall and returns to the infeed point.
4 Steps 2 and 3 are repeated until the programmed milling depth Q1 is reached.
5 If you have defined the tolerance in Q21, the TNC then remachines the slot walls to be as parallel as possible.
6 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on machine parameter 7420).
Please note while programming:

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

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

The memory capacity for programming an SL cycle is limited. You can program up to 8192 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 (DIN 844).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

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

- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range: -99999.9999 to 99999.9999

- **Finishing allowance for side** Q3 (incremental): Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999, alternatively PREDEF

- **Plunging depth** Q10 (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ

- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ

- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999

- **Dimension type? ang./lin.** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

- **Slot width** Q20: Width of the slot to be machined. Input range -99999.9999 to 99999.9999

- **Tolerance?** Q21: If you use a tool smaller than the programmed slot width Q20, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance Q21, the TNC adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With Q21 you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the remachining takes. **Recommendation**: Use a tolerance of 0.02 mm. **Function inactive**: Enter 0 (default setting) Input range 0 to 9.9999

**Example: NC blocks**

```
63 CYCL def 28 CYLINDER SURFACE
Q1=-8 ;MILLING DEPTH
Q3=+0 ;ALLOWANCE FOR SIDE
Q6=+0 ;SETUP CLEARANCE
Q10=+3 ;PLUNGING DEPTH
Q11=100 ;FEED RATE FOR PLUNGING
Q12=350 ;FEED RATE FOR MILLING
Q16=25 ;RADIUS
Q17=0 ;TYPE OF DIMENSION
Q20=12 ;SLOT WIDTH
Q21=0 ;TOLERANCE
```
8.4 CYLINDER SURFACE Ridge Milling (Cycle 29, DIN/ISO: G129, Software Option 1)

Cycle run

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

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

1 The TNC positions the tool over the starting point of machining. The TNC calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins from the left (1, RL = climb milling) or the right of the ridge (2, RR = up-cut milling).

2 After the TNC has positioned to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the ridge wall. If so programmed, it will leave metal for the finishing allowance.

3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate Q12 until the stud is completed.

4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.

5 Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.

6 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on machine parameter 7420).
Please note while programming:

- The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder. Refer to your machine tool manual.

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

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

The memory capacity for programming an SL cycle is limited. You can program up to 8192 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.

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

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

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

Example: NC blocks

```
63 CYCL DEF 29 CYLINDER SURFACE RIDGE
Q1=-8 ; MILLING DEPTH
Q3=+0 ; ALLOWANCE FOR SIDE
Q6=+0 ; SETUP CLEARANCE
Q10=+3 ; PLUNGING DEPTH
Q11=100 ; FEED RATE FOR PLUNGING
Q12=350 ; FEED RATE FOR MILLING
Q16=25 ; RADIUS
Q17=0 ; TYPE OF DIMENSION
Q20=12 ; RIDGE WIDTH
```
8.5 CYLINDER SURFACE Outside Contour Milling (Cycle 39, DIN/ISO: G139, Software Option 1)

Cycle run

This cycle enables you to program an open contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. 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.

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 (standard behavior).

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 contour. If so programmed, it will leave metal for the finishing allowance.

3. At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12 until the contour train is completed.

4. The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.

5. Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.

6. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on machine parameter 7420).

You can define the approach behavior of Cycle 39 in Machine Parameter 7680, bit 16.

- Bit 16 = 0: Tangential approach and departure
- Bit 16 = 1: Move to depth vertically at the starting point of the contour without tangential tool approach and move up at the contour end point without tangential departure.
Please note while programming:

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

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

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

The memory capacity for programming an SL cycle is limited. You can program up to 8192 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.

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.
8.5 CYLINDER SURFACE Outside Contour Milling (Cycle 39, DIN/ISO: G139, Software Option 1)

**Cycle parameters**

- **Milling depth Q1** (incremental): Distance between the cylindrical surface and the floor of the contour. Input range: -99999.9999 to 99999.9999

- **Finishing allowance for side Q3** (incremental): Finishing allowance on the contour wall. Input range: -99999.9999 to 99999.9999

- **Setup clearance Q6** (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999, alternatively PREDEF

- **Plunging depth Q10** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging Q11**: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ

- **Feed rate for milling Q12**: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ

- **Cylinder radius Q16**: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999

- **Dimension type? ang./lin. Q17**: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

**Example: NC blocks**

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

Example: Cylinder surface with Cycle 27

Note:
- Machine with B head and C table
- Cylinder centered on rotary table
- Datum at center of 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 on rotary table center
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0
   Positioning
TURN MBMAX FMAX
5 CYCL DEF 14.0 CONTOUR GEOMETRY
   Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1
7 CYCL DEF 27 CYLINDER SURFACE
   Define machining parameters
   Q1=-7 ; MILLING DEPTH
   Q3=+0 ; ALLOWANCE FOR SIDE
   Q6=2  ; SETUP CLEARANCE
   Q10=4 ; PLUNGING DEPTH
   Q11=100 ; FEED RATE FOR PLUNGING
   Q12=250 ; FEED RATE FOR MILLING
   Q16=25 ; RADIUS
   Q17=1 ; TYPE OF DIMENSION
```
8.6 Programming Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>L C+0 R0 FMAX M13 M99</td>
<td>Pre-position rotary table, spindle ON, call the cycle</td>
</tr>
<tr>
<td>9</td>
<td>L Z+250 R0 FMAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>10</td>
<td>PLANE RESET TURN FMAX</td>
<td>Tilt back, cancel the PLANE function</td>
</tr>
<tr>
<td>11</td>
<td>M2</td>
<td>End of program</td>
</tr>
<tr>
<td>12</td>
<td>LBL 1</td>
<td>Contour subprogram</td>
</tr>
<tr>
<td>13</td>
<td>L C+40 X+20 RL</td>
<td>Data for the rotary axis are entered in mm (Q17=1), traverse in the X axis because of 90° tilting</td>
</tr>
<tr>
<td>14</td>
<td>L C+50</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>RND R7.5</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>L X+60</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>RND R7.5</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>L IC-20</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>RND R7.5</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>L X+20</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>RND R7.5</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>L C+40</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>LBL 0</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>END PGM C27 MM</td>
<td></td>
</tr>
</tbody>
</table>
Example: Cylinder surface with Cycle 28

Notes:
- 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 I X+50 Y+0 R0 FMAX Position tool on rotary table center
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN FMAX Positioning
5 CYCL DEF 14.0 CONTOUR GEOMETRY Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1
7 CYCL DEF 28 CYLINDER SURFACE Define machining parameters
   Q1=-7 ;MILLING DEPTH
   Q3=+0 ;ALLOWANCE FOR SIDE
   Q6=2 ;SETUP CLEARANCE
   Q10=-4 ;PLUNGING DEPTH
   Q11=100 ;FEED RATE FOR PLNGNG
   Q12=250 ;FEED RATE FOR MILLING
   Q16=25 ;RADIUS
   Q17=1 ;TYPE OF DIMENSION
   Q20=10 ;SLOT WIDTH
   Q21=0.02 ;TOLERANCE Remachining active
<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>L C+0 R0 FMAX M3 M99</td>
<td>Pre-position rotary table, spindle ON, call the cycle</td>
</tr>
<tr>
<td>9</td>
<td>L Z+250 R0 FMAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>10</td>
<td>PLANE RESET TURN FMAX</td>
<td>Tilt back, cancel the PLANE function</td>
</tr>
<tr>
<td>11</td>
<td>M2</td>
<td>End of program</td>
</tr>
<tr>
<td>12</td>
<td>LBL 1</td>
<td>Contour subprogram, description of the midpoint path</td>
</tr>
<tr>
<td>13</td>
<td>L C+40 X+0 RL</td>
<td>Data for the rotary axis are entered in mm (Q17=1), traverse in the X axis because of 90° tilting</td>
</tr>
<tr>
<td>14</td>
<td>L X+35</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>L C+60 X+52.5</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>L X+70</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>LBL 0</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>END PGM C28 MM</td>
<td></td>
</tr>
</tbody>
</table>
Fixed Cycles: Contour Pocket with Contour Formula
9.1 SL Cycles with Complex Contour Formula

Basics

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.

Example: 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
```

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.
Properties of the subcontours

- By default, the TNC assumes that the contour is a pocket. Do not program a radius compensation. In the contour formula you can convert a pocket to an island by making it negative.
- 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. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the setup 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 for the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached on a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool 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.

With Machine Parameter 7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

Example: Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN PGM MODEL MM
1 DECLARE CONTOUR QC1 = "CIRCLE1"
2 DECLARE CONTOUR QC2 = "CIRCLE31XY"
3 DECLARE CONTOUR QC3 = "TRIANGLE"
4 DECLARE CONTOUR QC4 = "SQUARE"
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
...
...
```

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle 20.
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.
- Select the menu for functions for contour and point machining.
- Press the SEL CONTOUR soft key.
- Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the desired datum table.
- Select a program with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the SEL CONTOUR block.
- Conclude this function with the END key.
- Enter the full name of the program with the contour definition and confirm with the END key.

Alternatively you can also enter the program name or the complete path name of the program with the contour definition directly via the keyboard.

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.
- Select the menu for functions 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.
- Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the program to be called.
- Select the program with the contour description with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the SEL CONTOUR block.
- Define a separate depth for the selected contour.
- Conclude this function with the END key.

Alternatively you can also enter the name of the program with the contour description or the complete path name of the program directly via the keyboard.

With the given 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.
- Select the menu for functions for contour and point machining.
- Press the CONTOUR FORMULA soft key. The TNC then displays the following soft keys:

<table>
<thead>
<tr>
<th>Mathematical function</th>
<th>Soft key</th>
</tr>
</thead>
<tbody>
<tr>
<td>Intersected with</td>
<td></td>
</tr>
<tr>
<td>e.g. QC10 = QC1 &amp; QC5</td>
<td></td>
</tr>
<tr>
<td>Joined with</td>
<td></td>
</tr>
<tr>
<td>e.g. QC25 = QC7</td>
<td>QC18</td>
</tr>
<tr>
<td>Joined without intersection</td>
<td></td>
</tr>
<tr>
<td>e.g. QC12 = QC5 ^ QC25</td>
<td></td>
</tr>
<tr>
<td>Intersected with complement of</td>
<td></td>
</tr>
<tr>
<td>e.g. QC25 = QC1 \ QC2</td>
<td></td>
</tr>
<tr>
<td>Complement of the contour area</td>
<td></td>
</tr>
<tr>
<td>e.g. QC12 = #QC11</td>
<td></td>
</tr>
<tr>
<td>Opening parenthesis</td>
<td></td>
</tr>
<tr>
<td>e.g. QC12 = QC1 * (QC2 + QC3)</td>
<td></td>
</tr>
<tr>
<td>Closing parenthesis</td>
<td></td>
</tr>
<tr>
<td>e.g. QC12 = QC1 * (QC2 + QC3)</td>
<td></td>
</tr>
<tr>
<td>Defining a single contour</td>
<td></td>
</tr>
<tr>
<td>e.g. QC12 = QC1</td>
<td></td>
</tr>
</tbody>
</table>
Overlapping contours

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

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

Subprograms: overlapping pockets

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

Pockets A and B overlap.

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

The pockets are programmed as full circles.
Contour description program 1: pocket A

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

Contour description program 2: pocket B

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

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

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surfaces 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
Surface A is to be machined without the portion overlapped by B:
- The surfaces A and B must be entered in separate programs without radius compensation.
- In the contour formula, the surface B is subtracted from the surface A with the “joined with complement of” function.

Contour definition program:

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

Area of intersection
Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)
- The surfaces A and B must be entered in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the “intersection with” function.

Contour definition program:

```plaintext
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” on page 184).
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=+0.0 ; SURFACE COORDINATE
   Q6=0 ; SETUP CLEARANCE
   Q7=+100 ; CLEARANCE HEIGHT
   Q8=0.1 ; Rounding radius
   Q9=-1 ; DIRECTION
9 CYCL DEF 22 ROUGH-OUT  Cycle definition: Rough-out
   Q10=5 ; PLUNGING DEPTH
```
9.1 SL Cycles with Complex Contour Formula

Contour definition program with contour formula:

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM MODEL MM</td>
</tr>
<tr>
<td>1</td>
<td>DECLARE CONTOUR QC1 = &quot;CIRCLE1&quot;</td>
</tr>
<tr>
<td>2</td>
<td>FN 0: Q1 +=35</td>
</tr>
<tr>
<td>3</td>
<td>FN 0: Q2 = +50</td>
</tr>
<tr>
<td>4</td>
<td>FN 0: Q3 +=25</td>
</tr>
<tr>
<td>5</td>
<td>DECLARE CONTOUR QC2 = &quot;CIRCLE31XY&quot;</td>
</tr>
<tr>
<td>6</td>
<td>DECLARE CONTOUR QC3 = &quot;TRIANGLE&quot;</td>
</tr>
<tr>
<td>7</td>
<td>DECLARE CONTOUR QC4 = &quot;SQUARE&quot;</td>
</tr>
<tr>
<td>8</td>
<td>QC10 = ( QC 1</td>
</tr>
<tr>
<td>9</td>
<td>END PGM MODEL MM</td>
</tr>
</tbody>
</table>
### Contour description programs:

<table>
<thead>
<tr>
<th>0 BEGIN PGM CIRCLE1 MM</th>
<th>Contour description program: circle at right</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 CC X+65 Y+50</td>
<td></td>
</tr>
<tr>
<td>2 L PR+25 PA+0 RO</td>
<td></td>
</tr>
<tr>
<td>3 CP IPA+360 DR+</td>
<td></td>
</tr>
<tr>
<td>4 END PGM CIRCLE1 MM</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>0 BEGIN PGM CIRCLE31XY MM</th>
<th>Contour description program: circle at left</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 CC X+Q1 Y+Q2</td>
<td></td>
</tr>
<tr>
<td>2 LP PR+Q3 PA+0 RO</td>
<td></td>
</tr>
<tr>
<td>3 CP IPA+360 DR+</td>
<td></td>
</tr>
<tr>
<td>4 END PGM CIRCLE31XY MM</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>0 BEGIN PGM TRIANGLE MM</th>
<th>Contour description program: triangle at right</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 L X+73 Y+42 R0</td>
<td></td>
</tr>
<tr>
<td>2 L X+65 Y+58</td>
<td></td>
</tr>
<tr>
<td>3 L X+58 Y+42</td>
<td></td>
</tr>
<tr>
<td>4 L X+73</td>
<td></td>
</tr>
<tr>
<td>5 END PGM TRIANGLE MM</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>0 BEGIN PGM SQUARE MM</th>
<th>Contour description program: square at left</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 L X+27 Y+58 R0</td>
<td></td>
</tr>
<tr>
<td>2 L X+43</td>
<td></td>
</tr>
<tr>
<td>3 L Y+42</td>
<td></td>
</tr>
<tr>
<td>4 L X+27</td>
<td></td>
</tr>
<tr>
<td>5 L Y+58</td>
<td></td>
</tr>
<tr>
<td>6 END PGM SQUARE MM</td>
<td></td>
</tr>
</tbody>
</table>
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.

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. The secondary axes U,V,W are permitted.

Example: Program structure: Machining with SL cycles and complex contour formula

```plaintext
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
```
Characteristics of the fixed cycles

- The TNC automatically positions the tool to the setup 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 for the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached on a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool 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.

With Machine Parameter 7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

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

- Show the soft-key row with special functions.
- Select the menu for functions for contour and point machining.
- Press the CONTOUR DEF soft key. The TNC opens the dialog for entering the contour formula.
- Select the name of the first subcontour with the WINDOW SELECTION soft key or enter it directly. The first subcontour must always be the deepest pocket. Confirm with the ENT key.
- Specify via soft key whether the next subcontour is a pocket or an island. Confirm with the ENT key.
- Select the name of the second subcontour with the WINDOW SELECTION soft key or enter it directly. Confirm by pressing 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.

- As a rule, 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” on page 184).
9.2 SL Cycles with Simple Contour Formula

Fixed Cycles: Contour Pocket with Contour Formula
Fixed Cycles: Multipass Milling
10.1 Basics

Overview

The TNC offers four cycles for machining surfaces with the following characteristics:

- Created with a CAD/CAM system
- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 RUN 3-D DATA</td>
<td></td>
<td>Page 255</td>
</tr>
<tr>
<td>For multipass milling of 3-D data in several infeeds</td>
<td></td>
<td></td>
</tr>
<tr>
<td>230 MULTIPASS MILLING</td>
<td></td>
<td>Page 257</td>
</tr>
<tr>
<td>For flat rectangular surfaces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>231 RULED SURFACE</td>
<td></td>
<td>Page 259</td>
</tr>
<tr>
<td>For oblique, inclined or twisted surfaces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>232 FACE MILLING</td>
<td></td>
<td>Page 263</td>
</tr>
<tr>
<td>For level rectangular surfaces, with indicated oversizes and multiple infeeds</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
10.2 RUN 3-D DATA (Cycle 30, DIN/ISO: G60)

Cycle run

1 From the current position, the TNC positions the tool at rapid traverse $F_{\text{MAX}}$ in the tool axis to the setup clearance above the MAX point that you have programmed in the cycle.

2 The tool then moves at $F_{\text{MAX}}$ in the working plane to the MIN point you have programmed in the cycle.

3 From this point, the tool advances to the first contour point at the feed rate for plunging.

4 The TNC subsequently processes all points that are stored in the digitizing data file at the feed rate for milling. If necessary, the TNC retracts the tool between machining operations to the setup clearance if specific areas are to be left unmachined.

5 At the end of the cycle, the tool is retracted at $F_{\text{MAX}}$ to the setup clearance.

Please note while programming:

You can particularly use Cycle 30 to run conversational programs created offline in multiple infeeds.
Cycle parameters

- **PGM name 3-D data**: Enter the name of the program in which the contour data is stored. If the file is not stored in the current directory, enter the complete path. A maximum of 254 characters can be entered.

- **Min. point of range**: Lowest coordinates (X, Y and Z coordinates) in the range to be milled. Input range -99999.9999 to 99999.9999

- **Max. point of range**: Largest coordinates (X, Y and Z coordinates) in the range to be milled. Input range -99999.9999 to 99999.9999

- **Setup clearance 1**: Distance between tool tip and workpiece surface for tool movements at rapid traverse. Input range 0 to 99999.9999

- **Plunging depth 2**: Infeed per cut. Input range: -99999.9999 to 99999.9999

- **Feed rate for plunging 3**: Traversing speed of the tool during plunging in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO.

- **Feed rate for plunging 4**: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO.

- **Miscellaneous function M**: Optional entry of one to two miscellaneous functions, for example M13. Input range 0 to 999

Example: NC blocks

```
64 CYCL DEF 30.0 RUN 3-D DATA
65 CYCL DEF 30.1 PGM DIGIT.: BSP.H
66 CYCL DEF 30.2 X+0 Y+0 Z-20
67 CYCL DEF 30.3 X+100 Y+100 Z+0
68 CYCL DEF 30.4 SETUP 2
69 CYCL DEF 30.5 PECKG -5 F100
70 CYCL DEF 30.6 F350 M8
```
10.3 MULTIPASS MILLING (Cycle 230, DIN/ISO: G230)

Cycle run

1. From the current position in the working plane, the TNC positions the tool at rapid traverse $F_{\text{MAX}}$ to the starting point 1; the TNC moves the tool by its radius to the left and upward.

2. The tool then moves at $F_{\text{MAX}}$ in the tool axis to the setup clearance. From there it approaches the programmed starting position in the tool axis at the feed rate for plunging.

3. The tool then moves at the programmed feed rate for milling to the end point 2. The TNC calculates the end point from the programmed starting point, the program length, and the tool radius.

4. The TNC offsets the tool to the starting point in the next pass at the stepover feed rate. The offset is calculated from the programmed width and the number of cuts.

5. The tool then returns in the negative direction of the first axis.

6. Multipass milling is repeated until the programmed surface has been completed.

7. At the end of the cycle, the tool is retracted at $F_{\text{MAX}}$ to the setup clearance.

Please note while programming:

> From the current position, the TNC positions the tool at the starting point, first in the working plane and then in the spindle axis.

> Pre-position the tool in such a way that no collision between tool and clamping devices can occur.
Cycle parameters

- **Starting point in 1st axis** Q225 (absolute): Minimum point coordinate of the surface to be multipass-milled in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 2nd axis** Q226 (absolute): Minimum-point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 3rd axis** Q227 (absolute): Height in the spindle axis at which multipass-milling is carried out. Input range -99999.9999 to 99999.9999

- **First side length** Q218 (incremental): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in the 1st axis. Input range 0 to 99999.9999

- **Second side length** Q219 (incremental): Length of the surface to be multipass-milled in the minor axis of the working plane, referenced to the starting point in the 2nd axis. Input range 0 to 99999.9999

- **Number of cuts** Q240: Number of passes to be made over the width. Input range 0 to 99999

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving from setup clearance to the milling depth in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ.

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999; alternatively FAUTO, FU, FZ

- **Stepover feed rate** Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207. If you are moving it transversely in the open, Q209 may be greater than Q207. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ

- **Setup clearance** Q200 (incremental): Distance between tool tip and milling depth for positioning at the start and end of the cycle. Input range 0 to 99999.9999, alternatively PREDEF

---

**Example: NC blocks**

<table>
<thead>
<tr>
<th>71 CYCL DEF 230 MULTIPASS MILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q225=+10 ;STARTING PNT 1ST AXIS</td>
</tr>
<tr>
<td>Q226=+12 ;STARTING PNT 2ND AXIS</td>
</tr>
<tr>
<td>Q227=+2.5 ;STARTING PNT 3RD AXIS</td>
</tr>
<tr>
<td>Q218=150 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q219=75 ;SECOND SIDE LENGTH</td>
</tr>
<tr>
<td>Q240=25 ;NUMBER OF CUTS</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q209=200 ;STEPOVER FEED RATE</td>
</tr>
<tr>
<td>Q200=2 ;SETUP CLEARANCE</td>
</tr>
</tbody>
</table>
10.4 RULED SURFACE (Cycle 231, DIN/ISO: G231)

Cycle run

1 From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1.
2 The tool subsequently advances to the stopping point 2 at the feed rate for milling.
3 From this point, the tool moves at rapid traverse \( F_{\text{MAX}} \) by the tool diameter in the positive tool axis direction, and then back to starting point 1.
4 At the starting point 1 the TNC moves the tool back to the last traversed Z value.
5 Then the TNC moves the tool in all three axes from point 1 in the direction of point 4 to the next line.
6 From this point, the tool moves to the stopping point on this pass. The TNC calculates the end point from point 2 and a movement in the direction of point 3.
7 Multipass milling is repeated until the programmed surface has been completed.
8 At the end of the cycle, the tool is positioned above the highest programmed point in the spindle axis, offset by the tool diameter.
**Cutting motion**

The starting point, and therefore the milling direction, is selectable because the TNC always moves from point 1 to point 2 and in the total movement from point 1/2 to point 3/4. You can program point 1 at any corner of the surface to be machined.

If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

- A shaping cut (spindle axis coordinate of point 1 greater than spindle-axis coordinate of point 2) for slightly inclined surfaces.
- A drawing cut (spindle axis coordinate of point 1 smaller than spindle-axis coordinate of point 2) for steep surfaces.
- When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) parallel to the direction of the steeper inclination.

If you are using a spherical cutter for the machining operation, you can optimize the surface finish in the following way:

- When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) perpendicular to the direction of the steepest inclination.

**Please note while programming:**

From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1. Pre-position the tool in such a way that no collision between tool and fixtures can occur.

The TNC moves the tool with radius compensation R0 to the programmed positions.

If required, use a center-cut end mill (DIN 844).
10.4 RULED SURFACE (Cycle 231, DIN/ISO: G231)

Cycle parameters

- **Starting point in 1st axis** Q225 (absolute): Starting point coordinate of the surface to be multipass-milled in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 2nd axis** Q226 (absolute): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 3rd axis** Q227 (absolute): Starting point coordinate of the surface to be multipass-milled in the tool axis. Input range -99999.9999 to 99999.9999

- **2nd point in 1st axis** Q228 (absolute): End point coordinate of the surface to be multipass milled in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd point in 2nd axis** Q229 (absolute): End point coordinate of the surface to be multipass milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd point in 3rd axis** Q230 (absolute): End point coordinate of the surface to be multipass milled in the spindle axis. Input range -99999.9999 to 99999.9999

- **3rd point in 1st axis** Q231 (absolute): Coordinate of point 3 in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd point in 2nd axis** Q232 (absolute): Coordinate of point 3 in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd point in 3rd axis** Q233 (absolute): Coordinate of point 3 in the spindle axis. Input range: -99999.9999 to 99999.9999
10.4 RULED SURFACE (Cycle 231, DIN/ISO: G231)

- **4th point in 1st axis** Q234 (absolute): Coordinate of point 4 in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th point in 2nd axis** Q235 (absolute): Coordinate of point 4 in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th point in 3rd axis** Q236 (absolute): Coordinate of point 4 in the spindle axis. Input range -99999.9999 to 99999.9999
- **Number of cuts** Q240: Number of passes to be made between points 1 and 4, 2 and 3. Input range 0 to 99999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. The TNC performs the first step at half the programmed feed rate. Input range 0 to 99999.9999, alternatively FAUTO, FU, FZ

Example: NC blocks

```
72 CYCL DEF 231 RULED SURFACE
Q225=+0 ;STARTING PNT 1ST AXIS
Q226=+5 ;STARTING PNT 2ND AXIS
Q227=-2 ;STARTING PNT 3RD AXIS
Q228=+100 ;2ND POINT 1ST AXIS
Q229=+15 ;2ND POINT 2ND AXIS
Q230=+5 ;2ND POINT 3RD AXIS
Q231=+15 ;3RD POINT 1ST AXIS
Q232=+125 ;3RD POINT 2ND AXIS
Q233=+25 ;3RD POINT 3RD AXIS
Q234=+15 ;4TH POINT 1ST AXIS
Q235=+125 ;4TH POINT 2ND AXIS
Q236=+25 ;4TH POINT 3RD AXIS
Q240=40 ;NUMBER OF CUTS
Q207=500 ;FEED RATE FOR MILLING
```
10.5 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 within 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 \( F_{\text{MAX}} \) to the starting point 1 using positioning logic: If the current position in the spindle axis is greater than the 2nd setup clearance, the TNC positions the tool first in the machining plane and then in the spindle axis. Otherwise it first moves to the 2nd setup 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 then advances to the stopping point 2 at the 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 TNC retracts the tool at \( F_{\text{MAX}} \) to the 2nd setup clearance.


Strategy Q389=1

3 The tool then advances to the stopping point 2 at the feed rate for milling. The end point lies within the surface. The control 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 occurs within the workpiece borders.

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 TNC retracts the tool at FMAX to the 2nd setup clearance.

Strategy Q389=2

3 The tool then advances to the stopping point 2 at the feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.

4 The TNC positions the tool in the spindle axis to the setup 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 milling 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 TNC retracts the tool at FMAX to the 2nd setup clearance.

Please note while programming:

Enter the 2nd setup clearance in Q204 so that no collision between tool and clamping devices can occur.
## 10.5 FACE MILLING (Cycle 232, DIN/ISO: G232)

### Cycle parameters

- **Machining strategy (0/1/2) Q389**: Specify how the TNC is to machine the surface:
  - 0: Meander machining, stepover at positioning feed rate outside the surface to be machined
  - 1: Meander machining, stepover at feed rate for milling within the surface to be machined
  - 2: Line-by-line machining, retraction and stepover at the positioning feed rate

- **Starting point in 1st axis Q225** (absolute):
  Starting point coordinate of the surface to be machined in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 2nd axis Q226** (absolute):
  Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 3rd axis Q227** (absolute):
  Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999

- **End point in 3rd axis Q386** (absolute):
  Coordinate in the spindle axis to which the surface is to be face milled. Input range -99999.9999 to 99999.9999

- **First side length Q218** (incremental value):
  Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path in reference to the starting point in the 1st axis. Input range -99999.9999 to 99999.9999

- **Second side length Q219** (incremental value):
  Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the starting point in the 2nd axis. Input range -99999.9999 to 99999.9999
- **Maximum plunging depth** Q202 (incremental value): Maximum amount that the tool is advanced each time. The TNC calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range 0 to 99999.9999.

- **Allowance for floor** Q369 (incremental): Distance used for the last infeed. Input range 0 to 99999.9999.

- **Max. path overlap factor** Q370: Maximum stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. tooth radius when using a face-milling cutter), the TNC reduces the stepover accordingly. Input range 0.1 to 1.9999, alternatively **PREDEF**.

- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**.

- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min, while milling the last infeed. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**.

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207. Input range 0 to 99999.9999, alternatively **FMAX**, **FAUTO**, **PREDEF**.
Setup clearance Q200 (incremental): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the TNC moves the tool at the setup clearance over the current plunging depth to the starting point of the next pass. Input range 0 to 99999.9999, alternatively PREDEF

Clearance to side Q357 (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used. Input range 0 to 99999.9999

2nd setup clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999, alternatively PREDEF

Example: NC blocks

```
71 CYCL DEF 232 FACE MILLING
Q389=2 ;STRATEGY
Q225=+10 ;STARTING PNT 1ST AXIS
Q226=+12 ;STARTING PNT 2ND AXIS
Q227=+2.5 ;STARTING PNT 3RD AXIS
Q386=-3 ;END POINT IN 3RD AXIS
Q218=150 ;FIRST SIDE LENGTH
Q219=75 ;SECOND SIDE LENGTH
Q202=2 ;MAX. PLUNGING DEPTH
Q369=0.5 ;ALLOWANCE FOR FLOOR
Q370=1 ;MAX. OVERLAP
Q207=500 ;FEED RATE FOR MILLING
Q385=800 ;FEED RATE FOR FINISHING
Q253=2000 ;F PRE-POSITIONING
Q200=2 ;SETUP CLEARANCE
Q357=2 ;CLEARANCE TO SIDE
Q204=2 ;2ND SETUP CLEARANCE
```
Example: Multipass milling

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM C230 MM</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>BLK FORM 0.1 Z X+0 Y+0 Z+0</td>
<td>Definition of workpiece blank</td>
</tr>
<tr>
<td>2</td>
<td>BLK FORM 0.2 X+100 Y+100 Z+40</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>TOOL DEF 1 L+0 R+5</td>
<td>Tool definition</td>
</tr>
<tr>
<td>4</td>
<td>TOOL CALL 1 Z S3500</td>
<td>Tool call</td>
</tr>
<tr>
<td>5</td>
<td>L Z+250 R0 FMAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>6</td>
<td>CYCL DEF 230 MULTIPASS MILLING</td>
<td>Cycle definition: MULTIPASS MILLING</td>
</tr>
<tr>
<td></td>
<td>Q225=-0 ;STARTNG PNT 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q226=-0 ;STARTNG PNT 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q227=-35 ;STARTNG PNT 3RD AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q218=100 ;FIRST SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q219=100 ;2ND SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q240=25 ;NUMBER OF CUTS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q206250 ;FEED RATE FOR PLNGN</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q207=400 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q209=150 ;STEPOVER FEED RATE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q200=2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Line</td>
<td>Code</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------------</td>
</tr>
<tr>
<td>7</td>
<td>L X+25 Y0 R0 FMAX M3</td>
<td>Pre-position near the starting point</td>
</tr>
<tr>
<td>8</td>
<td>CYCL CALL</td>
<td>Cycle call</td>
</tr>
<tr>
<td>9</td>
<td>L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>10</td>
<td>END PGM C230 MM</td>
<td></td>
</tr>
</tbody>
</table>
11

Cycles: Coordinate Transformations
### 11.1 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:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7 DATUM SHIFT</td>
<td></td>
<td>Page 274</td>
</tr>
<tr>
<td>For shifting contours directly within the program or from datum tables</td>
<td></td>
<td></td>
</tr>
<tr>
<td>247 DATUM SETTING</td>
<td>247</td>
<td>Page 281</td>
</tr>
<tr>
<td>Datum setting during program run</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8 MIRROR IMAGE</td>
<td>6</td>
<td>Page 282</td>
</tr>
<tr>
<td>Mirroring contours</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10 ROTATION</td>
<td>10</td>
<td>Page 284</td>
</tr>
<tr>
<td>For rotating contours in the working plane</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11 SCALING FACTOR</td>
<td>11</td>
<td>Page 286</td>
</tr>
<tr>
<td>For increasing or reducing the size of contours</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26 AXIS-SPECIFIC SCALING FACTOR</td>
<td>26</td>
<td>Page 288</td>
</tr>
<tr>
<td>For increasing or reducing the size of contours with scaling factors for each axis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>19 WORKING PLANE</td>
<td>19</td>
<td>Page 290</td>
</tr>
<tr>
<td>Machining in tilted coordinate system on machines with swivel heads and/or rotary tables</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Effect of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM block (depending on MP7300).
- Select a new program
- Program miscellaneous function M142 "Erasing modal program information".
11.2 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.

Reset

- Program a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Use the TRANS DATUM RESET function.
- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.

Graphics

If you program a new BLK FORM after a datum shift, you can use MP 7310 to determine whether the BLK FORM is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.

Cycle parameters

- **Datum shift**: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to 6 NC axes, each from –99999.9999 to 99999.9999

Example: NC blocks

- 13 CYCL DEF 7.0 DATUM SHIFT
- 14 CYCL DEF 7.1 X+60
- 16 CYCL DEF 7.3 Z-5
- 15 CYCL DEF 7.2 Y+40
11.3 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.

Reset

- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.
- Execute a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition:
- Use the TRANS DATUM RESET function.

Graphics

If you program a new BLK FORM after a datum shift, you can use MP 7310 to determine whether the BLK FORM is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.

Status displays

In the additional status display, the following data from the datum table are shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number
Please note while programming:

**Danger of collision!**

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

MP7475, which earlier defined whether datums are referenced to the machine datum or the workpiece datum, now serves only as a safety measure. If MP7475 = 1, the TNC outputs an error message if a datum shift is called from a datum table.

Datum tables from the TNC 4xx whose coordinates are referenced to the machine datum (MP7475 = 1) cannot be used in the iTNC 530.

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 a program run mode to select the desired table for a 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.
Cycle parameters

- **Datum shift**: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number entered in the Q parameter. Input range: 0 to 9999

Selecting a datum table in the part program

With the **SEL TABLE** function you select the table from which the TNC takes the datums:

- To select the functions for program call, press the PGM CALL key.
- Press the DATUM TABLE soft key.
- Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the desired datum table.
- Select a datum table with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the **SEL TABLE** block.
- Conclude this function with the END key.

Alternatively you can also enter the table name or the complete path name of the table to be called directly via the keyboard.

Example: NC blocks

<table>
<thead>
<tr>
<th>NC block</th>
</tr>
</thead>
<tbody>
<tr>
<td>77 CYCL DEF 7.0 DATUM SHIFT</td>
</tr>
<tr>
<td>78 CYCL DEF 7.1 #5</td>
</tr>
</tbody>
</table>

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.

You can define datum tables and datum numbers in an NC block with the **TRANS DATUM TABLE** function.
Editing the datum table in the Programming and Editing 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 and Editing mode of operation.

- Press the PGM MGT key to call the file manager.
- Display the datum tables: Press the soft keys SELECT TYPE and SHOW .D.
- Select the desired table or enter a new file name.
- Edit the file. The soft-key row comprises the following functions for editing:

<table>
<thead>
<tr>
<th>Function</th>
<th>Soft key</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select beginning of table</td>
<td>BEGIN</td>
</tr>
<tr>
<td>Select end of table</td>
<td>END</td>
</tr>
<tr>
<td>Go to previous page</td>
<td>PAGE</td>
</tr>
<tr>
<td>Go to next page</td>
<td>PAGE</td>
</tr>
<tr>
<td>Insert line (only possible at end of table)</td>
<td>INSERT LINE</td>
</tr>
<tr>
<td>Delete line</td>
<td>DELETE LINE</td>
</tr>
<tr>
<td>Confirm the entered line and go to the beginning of the next line</td>
<td>NEXT LINE</td>
</tr>
<tr>
<td>Add the entered number of lines (datums) to the end of the table</td>
<td>APPEND N LINES</td>
</tr>
</tbody>
</table>
Editing a pocket table in a Program Run operating mode

In a program run mode you can select the active datum table. Press the DATUM TABLE soft key. You can then use the same editing functions as in the Programming and Editing mode of operation.

Transferring the actual values into the datum table

You can enter the current tool position or the last probed position in the datum table by pressing the “actual-position-capture” key:

► Place the text box on the line of the column in which you want to enter the position.

► Select the actual-position-capture function: The TNC opens a pop-up window that asks whether you want to enter the current tool position or the last probed values.

► Select the desired function with the arrow keys and confirm your selection with the ENT key.

► To enter the values in all axes, press the ALL VALUES soft key.

► To enter the value in the axis where the text box is located, press the CURRENT VALUE soft key.
### Configuring the datum table

In the second and third soft-key rows you can define for each datum table the axes for which you wish to set the datums. In the standard setting all of the axes are active. If you wish to exclude an axis, set the corresponding soft key to OFF. The TNC then deletes that column from the datum table.

If you do not wish to define a datum table for an active axis, press the NO ENT key. The TNC then enters a dash in that column.

### To exit a datum table

Select a different type of file in file management and choose the desired file.
11.4 DATUM SETTING (Cycle 247, DIN/ISO: G247)

Effect

With the Cycle DATUM SETTING, 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 active datum shift.

The TNC sets the preset only in the axes that are defined with values in the preset table. The datums of axes marked with – remain unchanged.

If you activate preset number 0 (line 0), then you activate the datum that you last set in a manual operating mode.

Cycle 247 is not functional in Test Run mode.

Example: NC blocks

```plaintext
13 CYCL DEF 247 DATUM SETTING
Q339=4 ;DATUM NUMBER
```
11.5 MIRROR IMAGE (Cycle 8, DIN/ISO: G28)

Effect

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

The mirror image cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction is reversed. (Except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

The result of the mirror image 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.

Reset

Program the MIRROR IMAGE cycle once again with NO ENT.

Please note while programming:

If you mirror only one axis, the machining direction is reversed for the milling cycles (Cycles 2xx). Exception: Cycle 208, in which the direction defined in the cycle applies.
**Cycle parameters**

- **Mirrored axis?**: Enter the axis to be mirrored. You can mirror all axes except for the spindle axis—including rotary axes—with the exception of the spindle axis and its associated auxiliary axis. You can enter up to three axes. Input range: Up to three NC axes X, Y, Z, U, V, W, A, B, C.

**Example: NC blocks**

- CYCL DEF 8.0 MIRROR IMAGE
- CYCL DEF 8.1 X Y U
11.6 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

Reset

Program the ROTATION cycle once again with a rotation angle of 0°.

Please note while programming:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.
**Cycle parameters**

- **Rotation**: Enter the rotation angle in degrees (°). Input range –360.000° to +360.000° (absolute or incremental)

**Example: 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
```
11.7 SCALING (Cycle 11, DIN/ISO: G72)

Effect

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- The working plane, or on all three coordinate axes at the same time (depending on MP 7410)
- Dimensions in cycles
- The parallel axes U,V,W

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 and up to 99.999 999
Reduction: SCL less than 1 and at least 0.000 001

Reset

Program the SCALING FACTOR cycle once again with a scaling factor of 1.
**Cycle parameters**

- **Scaling factor?**: Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under “Effect” above). Input range: 0.000000 to 99.999999

---

**Example: 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
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1
```
11.8 AXIS-SPECIFIC SCALING (Cycle 26)

Effect

With Cycle 26 you can account for shrinkage and oversize factors for each axis.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

Reset

Program the SCALING FACTOR cycle once again with a scaling factor of 1 for the same axis.

Please note while programming:

- Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.
- You can program each coordinate axis with its own axis-specific scaling factor.
- In addition, you can enter the coordinates of a center for all scaling factors.
- The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.
Cycle parameters

- **Axis and scaling factor**: Select the coordinate axis/axes by soft key and enter the factor(s) involved in enlarging or reducing. Input range: 0.000000 to 99.999999

- **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range: -99999.9999 to 99999.9999

Example: 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
```
11.9 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).

Warning!

Depending on your machine configuration, two mathematical solutions (axis positions) are possible for a spatial angle definition. Conduct appropriate tests on your machine to find out which axis position the TNC software selects in each case.

If the DCM software option is available to you, the axis position can be displayed in the PROGRAM + KINEMATICS view during test run (see User’s Manual for Conversational Programming, Dynamic Collision Monitoring).
The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You must move all axes to activate compensation for all axes.

If you set the function **Tilting program run** to **Active** in the Manual Operation mode, the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.

**Please note while programming:**

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the rotary axes or as mathematical angles of a tilted plane. Refer to your machine tool 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.

**Danger of collision!**

Ensure that the last defined angle is smaller than 360°.
Cycle parameters

- **Rotary axis and tilt angle?**: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range: -360.000 to 360.000

If the TNC automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate? F=**: Traverse speed of the rotary axis during automatic positioning. Input range: 0 to 99999.999
- **Setup clearance?** (incremental value): The TNC positions the tilting head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range: 0 to 99999.9999

**Danger of collision!**

Please note that the setup clearance in Cycle 19 does not refer to the upper edge of the workpiece (as is the case in the fixed cycles) but rather to the active datum.

**Resetting**

To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle once again by answering the dialog question with the NO ENT key to disable the function.
Positioning the axes of rotation

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

Manual positioning of rotary axes

If the rotary axes are not positioned automatically in Cycle 19, you must position them in a separate L block after the cycle definition.

If you use axis angles, you can define the axis values right in the L block. If you use spatial angles, then use the Q parameters Q120 (A-axis value), Q121 (B-axis value) and Q122 (C-axis value), which are described by Cycle 19.

Example NC blocks:

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 L Z+100 R0 FMAX</td>
<td>Define the spatial angle for calculation of the compensation</td>
</tr>
<tr>
<td>11 L X+25 Y+10 R0 FMAX</td>
<td>Position the rotary axes by using values calculated by Cycle 19</td>
</tr>
<tr>
<td>12 CYCL DEF 19.0 WORKING PLANE</td>
<td>Activate compensation for the spindle axis</td>
</tr>
<tr>
<td>13 CYCL DEF 19.1 A+0 B+45 C+0</td>
<td>Activate compensation for the working plane</td>
</tr>
<tr>
<td>14 L A+Q120 C+Q122 R0 F1000</td>
<td>Position the rotary axes by using values calculated by Cycle 19</td>
</tr>
<tr>
<td>15 L Z+80 R0 FMAX</td>
<td>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.</td>
</tr>
<tr>
<td>16 L X-8.5 Y-10 R0 FMAX</td>
<td></td>
</tr>
</tbody>
</table>

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.
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 setup 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:

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>L Z+100 R0 FMAX</td>
</tr>
<tr>
<td>11</td>
<td>L X+25 Y+10 R0 FMAX</td>
</tr>
<tr>
<td>12</td>
<td>CYCL DEF 19.0 WORKING PLANE</td>
</tr>
</tbody>
</table>
| 13   | CYCL DEF 19.1 A+0 B+45 C+0 F5000 SETUP50 | Define the angle for calculation of the compensation
| 14   | L Z+80 R0 FMAX | Also define the feed rate and the clearance
| 15   | L X-8.5 Y-10 R0 FMAX | Activate compensation for the spindle axis
|      |          | Activate compensation for the working plane
Position display in the tilted system

On activation of Cycle 19, the displayed positions (ACTL and NOML) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If necessary, the TNC outputs an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system.

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92) can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.
Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

1st: Activate the datum shift
2nd: Activate tilting function
3rd: Activate rotation
...
Machining
...
1st: Reset the rotation
2nd: Reset the tilting function
3rd: Reset the datum shift

Automatic workpiece measurement in the tilted system

The TNC measuring cycles enable you to have the TNC measure a workpiece in a tilted system automatically. The TNC stores the measured data in Q parameters for further processing (for example, for printout).
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 (clamping devices) during tilting.
   ▶ If required, position the rotary axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).
   ▶ Activate datum shift if required.
   ▶ Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes.
   ▶ Traverse all principal axes (X, Y, Z) to activate compensation.
   ▶ Write the program as if the machining process were to be executed in a non-tilted plane.
   ▶ If required, define Cycle 19 WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle 19. You can define the new angular values directly.
   ▶ Reset Cycle 19 WORKING PLANE; program 0° for all rotary axes.
   ▶ Disable the WORKING PLANE function; redefine Cycle 19 and answer the dialog question with NO ENT.
   ▶ Reset datum shift if required.
   ▶ Position the rotary axes to the 0° position, if required.

2 Clamp the workpiece

3 Preparations in the operating mode
   Positioning with Manual Data Input (MDI)
   Pre-position the rotary axis/axes to the corresponding angular value(s) for setting the datum. The angular value depends on the selected reference plane on the workpiece.
4 Preparations in the operating mode
   Manual Operation

   Use the 3D-ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the Manual Operating mode. For open loop axes, enter the angular values for the rotary axes into the menu.

   If the axes are not controlled, the angular values entered in the menu must correspond to the actual position(s) of the rotary axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Datum setting
   - Manually by touching the workpiece with the tool in the untilted coordinate system.
   - 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).

6 Start the part program in the operating mode Program Run, Full Sequence

7 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.
11.10 Programming Examples

Example: Coordinate transformation cycles

Program sequence
- Program the coordinate transformations in the main program
- Machining within a subprogram

Program sequence:

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN</td>
<td>PGM COTRANS MM</td>
</tr>
<tr>
<td>1</td>
<td>BLK</td>
<td>FORM 0.1 Z X+0 Y+0 Z-20</td>
</tr>
<tr>
<td>2</td>
<td>BLK</td>
<td>FORM 0.2 X+130 Y+130 Z+0</td>
</tr>
<tr>
<td>3</td>
<td>TOOL</td>
<td>DEF 1 L+0 R+1</td>
</tr>
<tr>
<td>4</td>
<td>TOOL</td>
<td>CALL 1 Z S4500</td>
</tr>
<tr>
<td>5</td>
<td>L</td>
<td>Z+250 R0 FMAX</td>
</tr>
<tr>
<td>6</td>
<td>CYCL</td>
<td>DEF 7.0 DATUM SHIFT</td>
</tr>
<tr>
<td>7</td>
<td>CYCL</td>
<td>DEF 7.1 X+65</td>
</tr>
<tr>
<td>8</td>
<td>CYCL</td>
<td>DEF 7.2 Y+65</td>
</tr>
<tr>
<td>9</td>
<td>CALL</td>
<td>LBL 1</td>
</tr>
<tr>
<td>10</td>
<td>LBL</td>
<td>10</td>
</tr>
<tr>
<td>11</td>
<td>CYCL</td>
<td>DEF 10.0 ROTATION</td>
</tr>
<tr>
<td>12</td>
<td>CYCL</td>
<td>DEF 10.1 IROT+45</td>
</tr>
<tr>
<td>13</td>
<td>CALL</td>
<td>LBL 1</td>
</tr>
<tr>
<td>14</td>
<td>CALL</td>
<td>LBL 10 REP 6/6</td>
</tr>
<tr>
<td>15</td>
<td>CYCL</td>
<td>DEF 10.0 ROTATION</td>
</tr>
<tr>
<td>16</td>
<td>CYCL</td>
<td>DEF 10.1 ROT+0</td>
</tr>
<tr>
<td>17</td>
<td>TRANS</td>
<td>DATUM RESET</td>
</tr>
</tbody>
</table>

Definition of workpiece blank
Tool definition
Tool call
Retract the tool
Shift datum to center
Call milling operation
Set label for program section repeat
Rotate by 45° (incremental)
Call milling operation
Return jump to LBL 10; repeat the milling operation six times
Reset the rotation
Reset the datum shift
<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>19</td>
<td>LBL 1</td>
<td>Subprogram 1</td>
</tr>
<tr>
<td>20</td>
<td>L X+0 Y+0 R0 FMAX</td>
<td>Define milling operation</td>
</tr>
<tr>
<td>21</td>
<td>L Z+2 R0 FMAX M3</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>L Z-5 R0 F200</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>L X+30 RL</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>L IY+10</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>RND R5</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>L IX+20</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>L IX+10 IY-10</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>RND R5</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>L IX-10 IY-10</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>L IX-20</td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>L IY+10</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>L X+0 Y+0 R0 F5000</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>L Z+20 R0 FMAX</td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>LBL 0</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>END PGM COTRANS MM</td>
<td></td>
</tr>
</tbody>
</table>
Cycles: Special Functions
### 12.1 Fundamentals

#### Overview

The TNC provides four cycles for the following special purposes:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 DWELL TIME</td>
<td><img src="clock.png" alt="" /></td>
<td>Page 303</td>
</tr>
<tr>
<td>12 PROGRAM CALL</td>
<td><img src="ppcm.png" alt="" /></td>
<td>Page 304</td>
</tr>
<tr>
<td>13 ORIENTED SPINDLE STOP</td>
<td><img src="spindle.png" alt="" /></td>
<td>Page 306</td>
</tr>
<tr>
<td>32 TOLERANCE</td>
<td><img src="tolerance.png" alt="" /></td>
<td>Page 307</td>
</tr>
</tbody>
</table>
12.2 DWELL TIME (Cycle 9, DIN/ISO: G04)

Function

This causes the execution of the next block within a running program to be delayed by the programmed DWELL TIME. A dwell time can be used for such purposes as chip breaking.

The cycle becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.

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.

Example: NC blocks

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>89</td>
<td>CYCL DEF 9.0 DWELL TIME</td>
</tr>
<tr>
<td>90</td>
<td>CYCL DEF 9.1 DWELL 1.5</td>
</tr>
</tbody>
</table>
12.3 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 and then called like fixed cycles.

Please note while programming:

- The program you are calling must be stored on the hard disk 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 to enter the program name.
- If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example TNC:KLAR35\FK1\50.H.
- If you want to define a DIN/ISO program to be a cycle, enter the file type .I behind the program name.
- As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called program can also influence the calling program.
Cycle parameters

- **Program name**: Enter the name of the program you want to call and, if necessary, the directory it is located in. A maximum of 254 characters can be entered.

The following functions can be used to call the defined program:

- CYCL CALL (separate block), or
- CYCL CALL POS (separate block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Example: Designate program 50 as a cycle and call it with M99

```
55 CYCL DEF 12.0 PGM CALL
56 CYCL DEF 12.1 PGM TNC:\KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99
```
12.4 ORIENTED SPINDLE STOP  
(Cycle 13, DIN/ISO: G36)

Cycle function

The TNC can control the machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for
- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set by the machine manufacturer (see your machine manual).

Please note while programming:

Cycle 13 is used internally for machining 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°

Example: NC blocks

```
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180
```
12.5 TOLERANCE (Cycle 32, DIN/ISO: G62)

Cycle function

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 smoothes 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. You can change the pre-set tolerance value with Cycle 32.

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle. The cycle may be locked.
Influences of the geometry definition in the CAM system

The most important factor of influence in offline NC program creation is the chord error $S$ defined in the CAM system. The maximum point spacing of NC programs generated in a postprocessor (PP) is defined through the chord error. If the chord error is less than or equal to the tolerance value $T$ defined in Cycle 32, then the TNC can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing if in Cycle 32 you choose a tolerance value between 110% and 200% of the CAM chord error.
Please note while programming:

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the TNC, but by the fact that, in order to machine the contour element transitions very exactly, the TNC might have to drastically reduce the speed.

Cycle 32 is DEF active which means that it becomes effective as soon as it is defined in the part program.

The TNC resets Cycle 32 if you

- Redefine it and confirm the dialog question for the **tolerance value** with NO ENT
- Select a new program with the PGM MGT key.

After you have reset Cycle 32, the TNC reactivates the tolerance that was predefined by machine parameter.

In a program with millimeters set as unit of measure, the TNC interprets the entered tolerance value in millimeters. In an inch program it interprets it as inches.

If you transfer a program with Cycle 32 that contains only the cycle parameter **Tolerance value** T, the control inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases. If the HSC filter is active on your machine (ask your machine manufacturer, if necessary), the circle can also become larger.

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 (**TCPM FUNCTION**). 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

**Example: NC blocks**

```
95 CYCL DEF 32.0 TOLERANCE
96 CYCL DEF 32.1 T0.05
97 CYCL DEF 32.2 HSC MODE:1 TA5
```

The HSC MODE and TA parameters are only available if on your machine you have software option 2 active (HSC machining).
Using Touch Probe Cycles
13.1 General Information about Touch Probe Cycles

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. The machine tool manual provides further information.

If you are carrying out measurements during program run, be sure that the tool data (length, radius) can be used from the calibrated data or from the last TOOL CALL block (selected with MP7411).

Method of function

Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder determines the probing feed rate in a machine parameter (see “Before You Start Working with Touch Probe Cycles” later in this chapter).

When the probe stylus contacts the workpiece,
- the 3-D touch probe transmits a signal to the TNC: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a distance defined in MP 6130, the TNC displays an error message.
Cycles in the Manual and El. Handwheel modes

In the Manual Operation and El. Handwheel modes, the TNC provides touch probe cycles that allow you to:
- Calibrate the touch probe
- Compensate workpiece misalignment
- Set datums

Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual and El. Handwheel modes, the TNC provides numerous cycles for a wide variety of applications in automatic mode:
- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Setting datums
- Automatic workpiece inspection
- Automatic tool measurement

You can program the touch probe cycles in the Programming and Editing operating mode via the TOUCH PROBE key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the TNC shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted (see figure at right).
13.1 General Information about Touch Probe Cycles

- The soft-key row shows all available touch probe functions divided into groups.
- Select the desired probe cycle, 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. 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.

### Example: 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 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=10 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+0 ;DATUM
```
13.2 Before You Start Working with Touch Probe Cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

**Maximum traverse to touch point: MP6130**

If the stylus is not deflected within the path defined in MP6130, the TNC outputs an error message.

**Safety clearance to touch point: MP6140**

In MP6140 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 must you define the touch point position. In many touch probe cycles you can also define a setup clearance in addition that is added to Machine Parameter 6140.

**Orient the infrared touch probe to the programmed probe direction: MP6165**

To increase measuring accuracy, you can use MP6165 = 1 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 MP6165, you must recalibrate the touch probe, because its deflection behavior changes.
Consider a basic rotation in the Manual Operation mode: MP6166

Set MP 6166 = 1 for the TNC to consider an active basic rotation during the probing process (the workpiece is approached along an angular path if required) to ensure that the measuring accuracy for probing individual positions is also increased in Setup mode.

Multiple measurement: MP6170

To increase measuring certainty, the TNC can run each probing process up to three times in sequence. If the measured position values differ too greatly, the TNC outputs an error message (the limit value is defined in MP6171). With multiple measurement it is possible to detect random errors, e.g. from contamination.

If the measured values lie within the confidence interval, the TNC saves the mean value of the measured positions.

Confidence interval for multiple measurement: MP6171

In MP6171 you store the value by which the results may differ when you make multiple measurements. If the difference in the measured values exceeds the value in MP6171, the TNC outputs an error message.

This feature is not active during the following functions in the Manual Operation mode:
- Calibrate length
- Calibrate radius
- Measure basic rotation
13.2 Before You Start Working with Touch Probe Cycles

**Touch trigger probe, probing feed rate: MP6120**

In MP6120 you define the feed rate at which the TNC is to probe the workpiece.

**Touch trigger probe, rapid traverse for positioning: MP6150**

In MP6150 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: MP6151**

In MP6151 you define whether the TNC is to position the touch probe at the feed rate defined in MP6150 or at rapid traverse.

- **Input value = 0:** Position at feed rate from MP6150
- **Input value = 1:** Pre-position at rapid traverse

**KinematicsOpt: Tolerance limit in Optimization mode: MP6600**

In **MP6600** you define the tolerance limit starting from which the TNC displays a note in the Optimizing mode when the measured kinematic data is greater than this limit value. The default value is 0.05. The larger the machine, the greater these values should be.

- **Input range:** 0.001 to 0.999

**KinematicsOpt, permissible deviation of the calibration ball radius: MP6601**

In **MP6601** you define the maximum permissible deviation from the entered cycle parameter by the calibration ball radius measured in the cycles.

- **Input range:** 0.01 to 0.1

The TNC calculates the calibration ball radius twice at every measuring point for all 5 touch points. If the radius is greater than Q407 + MP6601 an error message appears because it could be contamination.

If the radius found by the TNC is less than 5 * (Q407 - MP6601), the TNC also issues an error message.
13.2 Before You Start Working with Touch Probe Cycles

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.

Make sure that at the beginning of the cycle the compensation data (length, radius) from the calibrated data or from the last TOOL CALL block are active (selection via MP7411, see the User’s Manual of the iTNC530, “General User Parameters”).

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 south pole of the stylus is greater than the coordinate of the clearance height, the TNC first positions the probe in the working plane to the first starting position and then moves it immediately to the measuring height in the touch probe axis.
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment
14.1 Fundamentals

Overview

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:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>400 BASIC ROTATION</td>
<td><img src="image1.png" alt="Image" /></td>
<td>Page 322</td>
</tr>
<tr>
<td>401 ROT OF 2 HOLES</td>
<td><img src="image2.png" alt="Image" /></td>
<td>Page 325</td>
</tr>
<tr>
<td>402 ROT OF 2 STUDS</td>
<td><img src="image3.png" alt="Image" /></td>
<td>Page 328</td>
</tr>
<tr>
<td>403 ROT IN ROTARY AXIS</td>
<td><img src="image4.png" alt="Image" /></td>
<td>Page 331</td>
</tr>
<tr>
<td>405 ROT IN C AXIS</td>
<td><img src="image5.png" alt="Image" /></td>
<td>Page 336</td>
</tr>
<tr>
<td>404 SET BASIC ROTATION</td>
<td><img src="image6.png" alt="Image" /></td>
<td>Page 335</td>
</tr>
</tbody>
</table>
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 \( \alpha \) (see figure at right). This enables you to measure the basic rotation against any straight line 1 of the workpiece and to establish the reference to the actual 0° direction 2.
14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

Cycle run

Touch probe cycle 400 determines a workpiece misalignment by measuring two points, which must lie on a straight surface. With the basic rotation function the TNC compensates the measured value.

1. Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the programmed probe starting point 1 at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

3. Then the touch probe moves to the next starting position 2 and probes the second position.

4. The TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC will reset an active basic rotation at the beginning of the cycle.
### Cycle parameters

**1st meas. point 1st axis** Q263 (absolute):
Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

**1st meas. point 2nd axis** Q264 (absolute):
Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

**2nd meas. point 1st axis** Q265 (absolute):
Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

**2nd meas. point 2nd axis** Q266 (absolute):
Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

**Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis

**Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
-1: Negative traverse direction
+1: Positive traverse direction

**Measuring height in the touch probe axis** Q261 (absolute):
Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

**Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

**Clearance height** Q260 (absolute):
Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF
Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Default setting for basic rotation Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000

Preset number in table Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. Input range 0 to 2999

Example: 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=+8 ;2ND POINT 2ND AXIS |
| Q272=2 ;MEASURING AXIS |
| Q267=+1 ;TRAVERSE DIRECTION |
| Q261=-5 ;MEASURING HEIGHT |
| Q320=0 ;SET-UP CLEARANCE |
| Q260=+20 ;CLEARANCE HEIGHT |
| Q301=0 ;MOVE TO CLEARANCE |
| Q307=0 ;PRESET BASIC ROTATION |
| Q305=0 ;NO. IN TABLE |
14.3 BASIC ROTATION from Two Holes (Cycle 401, DIN/ISO: G401)

Cycle run

The Touch Probe Cycle 401 measures the centers of two holes. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the two 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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) to the point entered as 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.

This touch probe cycle is not allowed when the tilted working plane function is active.

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
14.3 BASIC ROTATION from Two Holes (Cycle 401, DIN/ISO: G401)

- **1st hole: Center in 1st axis** Q268 (absolute): Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st hole: Center in 2nd axis** Q269 (absolute): Center of the first hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd hole: Center in 1st axis** Q270 (absolute): Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd hole: Center in 2nd axis** Q271 (absolute): Center of the second hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
**Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (Q402=1). In this case the misalignment is not saved as an angular value. Input range 0 to 2999.

**Basic rotation / alignment** Q402: Specify whether the TNC should compensate misalignment with a basic rotation, or by rotating the rotary table:
- 0: Set basic rotation
- 1: Rotate the rotary table
When you select rotary table, the TNC does not save the measured misalignment, not even when you have defined a table line in parameter Q305.

**Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
- 0: Do not reset the display of the rotary axis to 0 after alignment
- 1: Reset the display of the rotary axis to 0 after alignment
The TNC sets the display to 0 only if you have defined Q402=1.

Example: NC blocks

```
5 TCH PROBE 401 ROT OF 2 HOLES
Q268=+37 ;1ST CENTER IN 1ST AXIS
Q269=+12 ;1ST CENTER IN 2ND AXIS
Q270=+75 ;2ND CENTER IN 1ST AXIS
Q271=+20 ;2ND CENTER IN 2ND AXIS
Q261=-5 ;MEASURING HEIGHT
Q260=+20 ;CLEARANCE HEIGHT
Q307=0 ;PRESET BASIC ROTATION
Q305=0 ;NO. IN TABLE
Q402=0 ;ALIGNMENT
Q337=0 ;SET TO ZERO
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe in rapid traverse (value from MP6150) to the starting point 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 to the starting point for probing the second stud.

4. The TNC moves the touch probe 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.

This touch probe cycle is not allowed when the tilted working plane function is active.

If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X
Cycle parameters

- **1st stud: Center in 1st axis (absolute):** Center of the first stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st stud: Center in 2nd axis** Q269 (absolute): Center of the first stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Diameter of stud 1** Q313: Approximate diameter of the 1st stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999

- **Measuring height 1 in the probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 1 is to be measured. Input range -99999.9999 to 99999.9999

- **2nd stud: Center in 1st axis** Q270 (absolute): Center of the second stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd stud: Center in 2nd axis** Q271 (absolute): Center of the second stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Diameter of stud 2** Q314: Approximate diameter of the 2nd stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999

- **Measuring height 2 in the probe axis** Q315 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 2 is to be measured. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**
**Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:

0: Move at measuring height between measuring points
1: Move at clearance height between measuring points

Alternatively **PREDEF**

**Default setting for basic rotation** Q307

(absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000

**Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (Q402=1). In this case the misalignment is not saved as an angular value. Input range 0 to 2999

**Basic rotation / alignment** Q402: Specify whether the TNC should compensate misalignment with a basic rotation, or by rotating the rotary table:

0: Set basic rotation
1: Rotate the rotary table

When you select rotary table, the TNC does not save the measured misalignment, not even when you have defined a table line in parameter Q305.

**Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:

0: Do not reset the display of the rotary axis to 0 after alignment
1: Reset the display of the rotary axis to 0 after alignment

The TNC sets the display to 0 only if you have defined Q402=1.

---

**Example: NC blocks**

```
5 TCH PROBE 402 ROT OF 2 STUDS
Q268=-37 ;1ST CENTER IN 1ST AXIS
Q269=+12 ;1ST CENTER IN 2ND AXIS
Q313=60 ;DIAMETER OF STUD 1
Q261=-5 ;MEASURING HEIGHT 1
Q270=+75 ;2ND CENTER IN 1ST AXIS
Q271=+20 ;2ND CENTER IN 2ND AXIS
Q314=60 ;DIAMETER OF STUD 2
Q315=-5 ;MEASURING HEIGHT 2
Q320=0 ;SET-UP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q307=0 ;PRESET BASIC ROTATION
Q305=0 ;NO. IN TABLE
Q402=0 ;ALIGNMENT
Q337=0 ;SET TO ZERO
```
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 surface. 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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the programmed starting point 1 at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

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 moves the rotary axis, which was defined in the cycle, by the measured value. Optionally you can have the display set to 0 after alignment.
Please note while programming:

**Danger of collision!**

You can now also use Cycle 403 when the "Tilt the working plane" function is active. Ensure that the **clearance height** is sufficiently large so that no collisions can occur during the final positioning of the rotary axis.

The TNC does not check whether touch points and compensation axis match. This can result in compensation movements offset by 180°.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The order of the touch points influences the determined compensation angle. Ensure that the coordinate of touch point 1 in the axis perpendicular to the probing direction is smaller than the coordinate of touch point 2.

The TNC stores the measured angle in parameter **Q150**.
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF
**Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

**Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
- 0: Move at measuring height between measuring points
- 1: Move at clearance height between measuring points

**Axis for compensation motion** Q312: assignment of the rotary axis in which the TNC is to compensate the measured misalignment:
- 4: Compensate misalignment with rotary axis A
- 5: Compensate misalignment with rotary axis B
- 6: Compensate misalignment with rotary axis C

**Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
- 0: Do not reset the display of the rotary axis to 0 after alignment
- 1: Reset the display of the rotary axis to 0 after alignment

**Number in table** Q305: Enter the number in the preset table/datum table in which the TNC is to set the rotary axis to zero. Only effective if Q337 is set to 1. Input range 0 to 2999

**Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
- 0: Write the measured basic rotation as a datum shift in the active datum table. The reference system is the active workpiece coordinate system.
- 1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).

**Reference angle? (0=ref. axis)** Q380: Angle with which the TNC is to align the probed straight line. Only effective if the rotary axis C is selected (Q312=6). Input range -360.000 to 360.000

---

**Example: NC blocks**

```
5 TCH PROBE 403 ROT IN C-AXIS
Q263=+25 ;1ST POINT 1ST AXIS
Q264=+10 ;1ST POINT 2ND AXIS
Q265=+40 ;2ND POINT 1ST AXIS
Q266=+17 ;2ND POINT 2ND AXIS
Q272=2 ;MEASURING AXIS
Q267=+1 ;TRAVERSE DIRECTION
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SET-UP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q312=6 ;COMPENSATION AXIS
Q337=0 ;SET TO ZERO
Q305=1 ;NO. IN TABLE
Q303=+1 ;MEAS. VALUE TRANSFER
Q380=+0 ;REFERENCE ANGLE
```
14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)

Cycle run

With Touch Probe Cycle 404, you can set any basic rotation automatically during program run. This cycle is intended primarily for resetting a previous basic rotation.

Cycle parameters

- **Preset value for basic rotation**: Angular value at which the basic rotation is to be set. Input range -360.000 to 360.000

- **Number in table Q305**: Enter the number in the preset/datum table in which the TNC is to save the defined basic rotation. Input range 0 to 2999

Example: NC blocks

```
5 TCH PROBE 404 BASIC ROTATION
Q307=+0 ;PRESET BASIC ROTATION
Q305=1 ;NO. IN TABLE
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the starting point at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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 and probes the second touch point.

4. The TNC positions the probe to starting point and then to starting point to probe the third and fourth touch points and positions the touch probe on the hole centers 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.
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°.
14.7 Compensating Workpiece Misalignment by Rotating the C Axis (Cycle 405, DIN/ISO: G405)

**Cycle parameters**

- **Center in 1st axis** Q321 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q322 (absolute value): Center of the hole in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position (angle of the hole center). Input range -99999.9999 to 99999.9999

- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999

- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000

- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
Measuring height in the touch probe axis
Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999.

Setup clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF.

Clearance height Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF.

Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF.

Set to zero after alignment Q337: Definition of whether the TNC should set the display of the C axis to zero, or write the angular offset in column C of the datum table:
0: Set display of C to 0
>0: Write the angular misalignment, including algebraic sign, 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.

Example: NC blocks

```
5 TCH PROBE 405 ROT IN C AXIS
Q321=+50 ;CENTER IN 1ST AXIS
Q322=+50 ;CENTER IN 2ND AXIS
Q262=10 ;NOMINAL DIAMETER
Q325=+0 ;STARTING ANGLE
Q247=90 ;STEPPING ANGLE
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q337=0 ;SET TO ZERO
```
Example: Determining a basic rotation from two holes

![Diagram showing two holes and coordinates]

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM CYC401 MM</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>TOOL CALL 69 Z</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>TCH PROBE 401 ROT 2 HOLES</td>
<td></td>
</tr>
<tr>
<td>268</td>
<td>Q268=+25 ;1ST CENTER IN 1ST AXIS</td>
<td>Center of the 1st hole: X coordinate</td>
</tr>
<tr>
<td>269</td>
<td>Q269=+15 ;1ST CENTER IN 2ND AXIS</td>
<td>Center of the 1st hole: Y coordinate</td>
</tr>
<tr>
<td>270</td>
<td>Q270=+80 ;2ND CENTER IN 1ST AXIS</td>
<td>Center of the 2nd hole: X coordinate</td>
</tr>
<tr>
<td>271</td>
<td>Q271=+35 ;2ND CENTER IN 2ND AXIS</td>
<td>Center of the 2nd hole: Y coordinate</td>
</tr>
<tr>
<td>261</td>
<td>Q261=-5 ;MEASURING HEIGHT</td>
<td>Coordinate in the touch probe axis in which the measurement is made</td>
</tr>
<tr>
<td>260</td>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
<td>Height in the touch probe axis at which the probe can traverse without collision</td>
</tr>
<tr>
<td>307</td>
<td>Q307=+0 ;PRESET BASIC ROTATION</td>
<td>Angle of the reference line</td>
</tr>
<tr>
<td>402</td>
<td>Q402=1 ;ALIGNMENT</td>
<td>Compensate misalignment by rotating the rotary table</td>
</tr>
<tr>
<td>337</td>
<td>Q337=1 ;SET TO ZERO</td>
<td>Set the display to zero after the alignment</td>
</tr>
<tr>
<td>3</td>
<td>CALL PGM 35K47</td>
<td>Part program call</td>
</tr>
<tr>
<td>4</td>
<td>END PGM CYC401 MM</td>
<td></td>
</tr>
</tbody>
</table>
Touch Probe Cycles:
Automatic Datum Setting
## 15.1 Fundamentals

### Overview

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

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 345</td>
</tr>
<tr>
<td>409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 349</td>
</tr>
<tr>
<td>410 DATUM INSIDE RECTAN. Measuring the inside length and width of a rectangle, and defining the center as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 352</td>
</tr>
<tr>
<td>411 DATUM OUTSIDE RECTAN. Measuring the outside length and width of a rectangle, and defining the center as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 356</td>
</tr>
<tr>
<td>412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 360</td>
</tr>
<tr>
<td>413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 364</td>
</tr>
<tr>
<td>414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 368</td>
</tr>
<tr>
<td>415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as datum</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 373</td>
</tr>
<tr>
<td>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</td>
<td><img src="image" alt="Soft key" /></td>
<td>Page 377</td>
</tr>
</tbody>
</table>
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:

<table>
<thead>
<tr>
<th>Active touch probe axis</th>
<th>Datum setting in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z or W</td>
<td>X and Y</td>
</tr>
<tr>
<td>Y or V</td>
<td>Z and X</td>
</tr>
<tr>
<td>X or U</td>
<td>Y and Z</td>
</tr>
</tbody>
</table>
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 datum in the active datum table. The reference system is the active workpiece coordinate system. The value of parameter Q305 determines the datum number. **Activate datum with Cycle 7 in the part program.**

- **Q305 not equal to 0, Q303 = 1**
  The TNC writes the calculated datum in the preset table. The reference system is the machine coordinate system (REF coordinates). The value of parameter Q305 determines the preset number. **Activate 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 that are listed with every cycle description.
15.2 SLOT CENTER REF PT  
(Cycle 408, DIN/ISO: G408,  
FCL 3 Function)

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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the starting point at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

3 Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point 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 “Saving the calculated datum” on page 344) 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q166</td>
<td>Actual value of measured slot width</td>
</tr>
<tr>
<td>Q157</td>
<td>Actual value of the centerline</td>
</tr>
</tbody>
</table>
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.

**Cycle parameters**

- **Center in 1st axis** Q321 (absolute): Center of the slot in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q322 (absolute): Center of the slot in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Width of slot** Q311 (incremental): Width of the slot, regardless of its position in the working plane. Input range 0 to 99999.9999

- **Measuring axis (1=1st axis / 2=2nd axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternatively **PREDEF**

- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the slot center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the slot center. Input range 0 to 2999

- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated slot center. Default setting = 0. Input range: -99999.9999 to 99999.9999

- **Measured-value transfer** (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- 0: Do not set datum in the touch probe axis
- 1: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute):
Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute):
Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute):
Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

**Example: 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 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=10 ;NO. IN TABLE
Q405=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
```
15.3 RIDGE CENTER REF PT
(Cycle 409, DIN/ISO: G409,
FCL 3 function)

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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

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 “Saving the calculated datum” on page 344) 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q166</td>
<td>Actual value of measured ridge width</td>
</tr>
<tr>
<td>Q157</td>
<td>Actual value of the centerline</td>
</tr>
</tbody>
</table>

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.
Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the ridge in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Center in 2nd axis** Q322 (absolute): Center of the ridge in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Width of ridge** Q311 (incremental): Width of the ridge, regardless of its position in the working plane. Input range 0 to 99999.9999

- **Measuring axis (1=1st axis / 2=2nd axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis

- **Measuring height in the touch probe axis**
  Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999, alternatively **PREDEF**

- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the ridge center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the slot center. Input range 0 to 2999

- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated ridge center. Default setting = 0. Input range: -99999.9999 to 99999.9999
**Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
0: Do not set datum in the touch probe axis
1: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

---

**Example: NC blocks**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 409 SLOT CENTER RIDGE</td>
<td></td>
</tr>
<tr>
<td>Q321=+50 ;CENTER IN 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td>Q322=+50 ;CENTER IN 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td>Q311=25 ;RIDGE WIDTH</td>
<td></td>
</tr>
<tr>
<td>Q272=1 ;MEASURING AXIS</td>
<td></td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
<td></td>
</tr>
<tr>
<td>Q320=0 ;SETUP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td>Q305=10 ;NO. IN TABLE</td>
<td></td>
</tr>
<tr>
<td>Q405=+0 ;DATUM</td>
<td></td>
</tr>
<tr>
<td>Q303=+1 ;MEAS. VALUE TRANSFER</td>
<td></td>
</tr>
<tr>
<td>Q381=1 ;PROBE IN TS AXIS</td>
<td></td>
</tr>
<tr>
<td>Q382=+85 ;1ST CO. FOR TS AXIS</td>
<td></td>
</tr>
<tr>
<td>Q383=+50 ;2ND CO. FOR TS AXIS</td>
<td></td>
</tr>
<tr>
<td>Q384=+0 ;3RD CO. FOR TS AXIS</td>
<td></td>
</tr>
<tr>
<td>Q333=+1 ;DATUM</td>
<td></td>
</tr>
</tbody>
</table>
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

3. Then the touch probe moves either paraxially at the measuring height or linearly at the 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 “Saving the calculated datum” on page 344).

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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q154</td>
<td>Actual value of length in the reference axis</td>
</tr>
<tr>
<td>Q155</td>
<td>Actual value of length in the minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

Danger of collision!
To prevent a collision between touch probe and workpiece, enter low estimates for the lengths of the 1st and 2nd 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.

Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **First side length** Q323 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF
**Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
- **0**: Move at measuring height between measuring points
- **1**: Move at clearance height between measuring points
Alternatively **PREDEF**

**Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket.
Input range 0 to 2999

**New datum for reference axis** Q331 (absolute):
Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0.
Input range: –99999.9999 to 99999.9999

**New datum for minor axis** Q332 (absolute):
Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999

**Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
- **-1**: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
- **0**: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
- **1**: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- 0: Do not set datum in the touch probe axis
- 1: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

---

### Example: 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 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=10 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
```
15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411)

Cycle run

Touch Probe Cycle 411 finds the center of a rectangular stud and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point and probes the second touch point.

4. The TNC positions the probe to starting point and then to starting point 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 “Saving the calculated datum” on page 344).

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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q154</td>
<td>Actual value of length in the reference axis</td>
</tr>
<tr>
<td>Q155</td>
<td>Actual value of length in the minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

**Danger of collision!**

To prevent a collision between the 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.

**Cycle parameters**

- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **First side length** Q323 (incremental): Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**
Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Datum number in table Q305: Enter the datum number in the table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center. Input range 0 to 2999

New datum for reference axis Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999

New datum for minor axis Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
0: Do not set datum in the touch probe axis
1: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

---

**Example: NC blocks**

```
5 TCH PROBE 411 DATUM OUTSIDE 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 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=0 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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 “Saving the calculated datum” on page 344) 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q153</td>
<td>Actual value of diameter</td>
</tr>
</tbody>
</table>
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.

**Cycle parameters**

- **Center in 1st axis** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q322 (absolute): Center of the pocket in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999

- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999

- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000

- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) 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 -120.0000 to 120.0000
Measuring height in the touch probe axis Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

Setup clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

Clearance height Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF

Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Datum number in table Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket. Input range 0 to 2999

New datum for reference axis Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999

New datum for minor axis Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  - 0: Do not set datum in the touch probe axis
  - 1: Set datum in the touch probe axis

- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the hole with 4 or 3 probing points:
  - 4: Use 4 measuring points (standard setting)
  - 3: Use 3 measuring points

- **Type of traverse?** Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between the measuring points if “traverse to clearance height” (Q301=1) is active.
  - 0: Move between operations on a straight line
  - 1: Move between operations on the pitch circle

---

**Example: 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 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=12 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+0 ;DATUM
Q423=4 ;NO. OF MEAS. POINTS
Q365=1 ;TYPE OF TRAVERSE
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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 “Saving the calculated datum” on page 344) 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q153</td>
<td>Actual value of diameter</td>
</tr>
</tbody>
</table>
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°.

**Cycle parameters**

- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (- = 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 -120.0000 to 120.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999.

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**.

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**.

- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternatively **PREDEF**.

- **Datum number in table** Q305: Enter the datum number in the table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center. Input range 0 to 2999.

- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999.

- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999.

- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  - **-1**: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
  - **0**: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  - **1**: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

---

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- **0**: Do not set datum in the touch probe axis
- **1**: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

**No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
- **4**: Use 4 measuring points (standard setting)
- **3**: Use 3 measuring points

**Type of traverse?** Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between the measuring points if “traverse to clearance height” (Q301=1) is active.
- **0**: Move between operations on a straight line
- **1**: Move between operations on the pitch circle

---

**Example: NC blocks**

```plaintext
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
Q260=-20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q260-5 ;MEASURING HEIGHT
Q260+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=15 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
Q423=4 ;NO. OF MEAS. POINTS
Q365=1 ;TYPE OF TRAVERSE
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) to the first touch point 1 (see figure at upper right). The TNC offsets the touch probe by the safety clearance in the direction opposite the respective traverse direction.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed 3rd measuring point.

3. Then the touch probe moves to the next starting position 2 and probes the second position.

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 “Saving the calculated datum” on page 344) and saves the coordinates of the determined corner in the Q parameters listed below.

6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of corner in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of corner in minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

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 at lower right).

<table>
<thead>
<tr>
<th>Corner</th>
<th>X coordinate</th>
<th>Y coordinate</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Point 1 greater than point 3</td>
<td>Point 1 less than point 3</td>
</tr>
<tr>
<td>B</td>
<td>Point 1 less than point 3</td>
<td>Point 1 less than point 3</td>
</tr>
<tr>
<td>C</td>
<td>Point 1 less than point 3</td>
<td>Point 1 greater than point 3</td>
</tr>
<tr>
<td>D</td>
<td>Point 1 greater than point 3</td>
<td>Point 1 greater than point 3</td>
</tr>
</tbody>
</table>
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999

- **3rd meas. point 1st axis** Q296 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd meas. point 2nd axis** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**
Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Execute basic rotation Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
0: No basic rotation
1: Basic rotation

Datum number in table Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner. Input range 0 to 2999

New datum for reference axis Q331 (absolute):
Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0. Input range: –99999.9999 to 99999.9999

New datum for minor axis Q332 (absolute):
Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0. Input range: –99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- 0: Do not set datum in the touch probe axis
- 1: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

---

**Example: NC blocks**

```
5 TCH PROBE 414 DATUM INSIDE CORNER
Q263=+37 ;1ST POINT 1ST AXIS
Q264=+7 ;1ST POINT 2ND AXIS
Q326=50 ;SPACING IN 1ST AXIS
Q296=+95 ;3RD POINT 1ST AXIS
Q297=+25 ;3RD POINT 2ND AXIS
Q327=45 ;SPACING IN 2ND AXIS
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q304=0 ;BASIC ROTATION
Q305=7 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) to the first 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 the respective traverse direction.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The probing direction is derived from the number by which you identify the corner.

3 Then the touch probe moves to the next starting position and probes the second position.

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 “Saving the calculated datum” on page 344) and saves the coordinates of the determined corner in the Q parameters listed below.

6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of corner in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of corner in minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC always measures the first line in the direction of the minor axis of the working plane.

**Cycle parameters**

**1st meas. point 1st axis** Q263 (absolute):
Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

**1st meas. point 2nd axis** Q264 (absolute):
Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

**Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999

**Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999

**Corner** Q308: Number identifying the corner which the TNC is to set as datum. Input range 1 to 4

**Measuring height in the touch probe axis** Q261 (absolute):
Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

**Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

**Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF
Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Execute basic rotation Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
0: No basic rotation
1: Basic rotation

Datum number in table Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner. Input range 0 to 2999

New datum for reference axis Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0. Input range: –99999.9999 to 99999.9999

New datum for minor axis Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0. Input range: –99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:

0: Do not set datum in the touch probe axis
1: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

---

**Example: NC blocks**

```
5 TCH PROBE 415 DATUM OUTSIDE CORNER
Q263=+37 ;1ST POINT 1ST AXIS
Q264=+7 ;1ST POINT 2ND AXIS
Q326=50 ;SPACING IN 1ST AXIS
Q296=+95 ;3RD POINT 1ST AXIS
Q297=+25 ;3RD POINT 2ND AXIS
Q327=45 ;SPACING IN 2ND AXIS
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q304=0 ;BASIC ROTATION
Q305=7 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) to the point entered as center of the first hole.

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.

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.

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 “Saving the calculated datum” on page 344) 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q153</td>
<td>Actual value of bolt hole circle diameter</td>
</tr>
</tbody>
</table>
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Nominal diameter** Q262: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input range -0 to 99999.9999

- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000

- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000

- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF
**Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the bolt-hole circle center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the bolt hole center. Input range 0 to 2999

**New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the bolt-hole center. Default setting = 0. Input range: –99999.9999 to 99999.9999

**New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the bolt-hole center. Default setting = 0. Input range: –99999.9999 to 99999.9999

**Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- **0**: Do not set datum in the touch probe axis
- **1**: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

**Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140, and is only effective when the datum is probed in the touch probe axis. Input range 0 to 99999.9999, alternatively PREDEF

---

**Example: NC blocks**

```
5 TCH PROBE 416 DATUM CIRCLE CENTER
Q273=+50 ;CENTER IN 1ST AXIS
Q274=+50 ;CENTER IN 2ND AXIS
Q262=90 ;NOMINAL DIAMETER
Q291=+34 ;ANGLE OF 1ST HOLE
Q292=+70 ;ANGLE OF 2ND HOLE
Q293=+210 ;ANGLE OF 3RD HOLE
Q261=-5 ;MEASURING HEIGHT
Q260=+20 ;CLEARANCE HEIGHT
Q305=12 ;NO. IN TABLE
Q331=0 ;DATUM
Q332=0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
Q320=0 ;SETUP CLEARANCE
```
15.11 DATUM IN TOUCH PROBE AXIS  
(Cycle 417, DIN/ISO: G417)

Cycle run

Touch Probe Cycle 417 measures any coordinate in the touch probe axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.

1. Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the programmed probe starting point at rapid traverse (value from MP6150). 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 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 “Saving the calculated datum” on page 344) and saves the actual values in the Q parameters listed below.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q160</td>
<td>Actual value of measured point</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC then sets the datum in this axis.
**Cycle parameters**

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 3rd axis** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface. Input range 0 to 2999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  - **-1:** Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
  - **0:** Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  - **1:** Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

**Example: 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 ;SETUP CLEARANCE
Q260=+50 ;CLEARANCE HEIGHT
Q305=0 ;NO. IN TABLE
Q333=0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
```
15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)

Cycle run

Touch Probe Cycle 418 calculates the intersection of the lines connecting opposite holes and sets the datum at the intersection. If desired, the TNC can also enter the intersection into a datum table or preset table.

1. Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) 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 the 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 “Saving the calculated datum” on page 344). 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of intersection point in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of intersection point in minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

- **First center in 1st axis** Q268 (absolute): center of the 1st hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **First center in 2nd axis** Q269 (absolute): center of the 1st hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **First center in 1st axis** Q270 (absolute): center of the 2nd hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **First center in 2nd axis** Q271 (absolute): center of the 2nd hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **First center in 1st axis** Q316 (absolute): center of the 3rd hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd center in 2nd axis** Q317 (absolute): center of the 3rd hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **4th center in 1st axis** Q318 (absolute): center of the 4th hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **4th center in 2nd axis** Q319 (absolute): center of the 4th hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the line intersection. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines. Input range 0 to 2999.

- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0. Input range: -99999.9999 to 99999.9999.

- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0. Input range: -99999.9999 to 99999.9999.

- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  - **-1:** Do not use. Is entered by the TNC when old programs are read in (see “Saving the calculated datum” on page 344).
  - **0:** Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  - **1:** Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- **0**: Do not set datum in the touch probe axis
- **1**: Set datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute):
Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.

**Probe TS axis: Coord. 2nd axis** Q383 (absolute):
Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**Probe TS axis: Coord. 3rd axis** Q384 (absolute):
Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range –99999.9999 to 99999.9999

**New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

---

**Example: NC blocks**

```
5 TCH PROBE 418 DATUM FROM 4 HOLES
Q268=+20 ;1ST CENTER IN 1ST AXIS
Q269=+25 ;1ST CENTER IN 2ND AXIS
Q270=+150 ;2ND CENTER IN 1ST AXIS
Q271=+25 ;2ND CENTER IN 2ND AXIS
Q316=+150 ;3RD CENTER IN 1ST AXIS
Q317=+85 ;3RD CENTER IN 2ND AXIS
Q318=+22 ;4TH CENTER IN 1ST AXIS
Q319=+80 ;4TH CENTER IN 2ND AXIS
Q261=-5 ;MEASURING HEIGHT
Q260=+10 ;CLEARANCE HEIGHT
Q305=12 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+0 ;DATUM
```
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the programmed probe starting point 1 at rapid traverse (value from MP6150). 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 “Saving the calculated datum” on page 344).

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you use Cycle 419 several times in succession to save the datum in more than one axis in the preset table, you must activate the preset number last written to by Cycle 419 after every execution of Cycle 419 (this is not required if you overwrite the active preset).
## Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

- **Measuring axis (1..3: 1=reference axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

<table>
<thead>
<tr>
<th>Axis assignment</th>
<th>Active touch probe axis: Q272= 3</th>
<th>Corresponding reference axis: Q272 = 1</th>
<th>Corresponding minor axis: Q272 = 2</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Z</td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>Z</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>Y</td>
<td>Z</td>
</tr>
</tbody>
</table>
**Traverse direction** Q267: Direction in which the probe is to approach the workpiece:
- **-1:** Negative traverse direction
- **+1:** Positive traverse direction

**Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface. Input range 0 to 2999

**New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0. Input range: –99999.9999 to 99999.9999

**Measured-value transfer** (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
- **-1:** Do not use. See “Saving the calculated datum” on page 344
  - **0:** Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  - **1:** Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

---

**Example: NC blocks**

```
5 TCH PROBE 419 DATUM IN ONE AXIS
Q263=+25 ;1ST POINT 1ST AXIS
Q264=+25 ;1ST POINT 2ND AXIS
Q261=+25 ;MEASURING HEIGHT
Q320=0 ;SETUP CLEARANCE
Q260=+50 ;CLEARANCE HEIGHT
Q272=+1 ;MEASURING AXIS
Q267=+1 ;TRAVERSE DIRECTION
Q305=0 ;NO. IN TABLE
Q333=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
```
Example: Datum setting in center of a circular segment and on top surface of workpiece

```
O BEGIN PGM CYC413 MM
1 TOOL CALL 69 Z
```

Call tool 0 to define the touch probe axis
### 15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q321=+25</td>
<td>Center of circle: X coordinate</td>
</tr>
<tr>
<td>Q322=+25</td>
<td>Center of circle: Y coordinate</td>
</tr>
<tr>
<td>Q262=30</td>
<td>Circle diameter</td>
</tr>
<tr>
<td>Q325=+90</td>
<td>Polar coordinate angle for 1st touch point</td>
</tr>
<tr>
<td>Q247=+45</td>
<td>Stepping angle for calculating the starting points 2 to 4</td>
</tr>
<tr>
<td>Q261=-5</td>
<td>Coordinate in the touch probe axis in which the measurement is made</td>
</tr>
<tr>
<td>Q320=2</td>
<td>Safety clearance in addition to MP6140</td>
</tr>
<tr>
<td>Q260=+10</td>
<td>Height in the touch probe axis at which the probe can traverse without collision</td>
</tr>
<tr>
<td>Q301=0</td>
<td>Do not move to clearance height between measuring points</td>
</tr>
<tr>
<td>Q305=0</td>
<td>Set display</td>
</tr>
<tr>
<td>Q331=+0</td>
<td>Set the display in X to 0</td>
</tr>
<tr>
<td>Q332=+10</td>
<td>Set the display in Y to 10</td>
</tr>
<tr>
<td>Q303=+0</td>
<td>Without function, since display is to be set</td>
</tr>
<tr>
<td>Q381=1</td>
<td>Also set datum in the touch probe axis</td>
</tr>
<tr>
<td>Q382=+25</td>
<td>X coordinate of touch point</td>
</tr>
<tr>
<td>Q383=+25</td>
<td>Y coordinate of touch point</td>
</tr>
<tr>
<td>Q384=+25</td>
<td>Z coordinate of touch point</td>
</tr>
<tr>
<td>Q333=+0</td>
<td>Set the display in Z to 0</td>
</tr>
<tr>
<td>Q423=4</td>
<td>Number of measuring points</td>
</tr>
<tr>
<td>Q365=1</td>
<td>Position on circular arc or linearly to the next touch point</td>
</tr>
<tr>
<td>3 CALL PGM 35K47</td>
<td>Part program call</td>
</tr>
<tr>
<td>4 END PGM CYC413 MM</td>
<td>Part program call</td>
</tr>
</tbody>
</table>
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
2 TCH PROBE 417 DATUM IN TS AXIS
Q263=+7.5 ;1ST POINT 1ST AXIS
Q264=+7.5 ;1ST POINT 2ND AXIS
Q294=+25 ;1ST POINT 3RD AXIS
Q320=0 ;SETUP CLEARANCE
Q260=+50 ;CLEARANCE HEIGHT
Q305=1 ;NO. IN TABLE
Q333=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
```

Call tool 0 to define the touch probe axis
Cycle definition for datum setting in the touch probe axis
Touch point: X coordinate
Touch point: Y coordinate
Touch point: Z coordinate
Safety clearance in addition to MP6140
Height in the touch probe axis at which the probe can traverse without collision
Write Z coordinate in line 1
Set touch-probe axis to 0
In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)
### 15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Center of the bolt hole circle: X coordinate</td>
<td>Q273=+35</td>
</tr>
<tr>
<td>Center of the bolt hole circle: Y coordinate</td>
<td>Q274=+35</td>
</tr>
<tr>
<td>Diameter of the bolt hole circle</td>
<td>Q262=50</td>
</tr>
<tr>
<td>Polar coordinate angle for 1st hole center 1</td>
<td>Q291=+90</td>
</tr>
<tr>
<td>Polar coordinate angle for 2nd hole center 2</td>
<td>Q292=+180</td>
</tr>
<tr>
<td>Polar coordinate angle for 3rd hole center 3</td>
<td>Q293=+270</td>
</tr>
<tr>
<td>Coordinate in the touch probe axis in which the measurement is made</td>
<td>Q261=+15</td>
</tr>
<tr>
<td>Height in the touch probe axis at which the probe can traverse without collision</td>
<td>Q260=+10</td>
</tr>
<tr>
<td>Enter center of bolt hole circle (X and Y) in line 1</td>
<td>Q305=1</td>
</tr>
<tr>
<td>In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)</td>
<td>Q303=+1</td>
</tr>
<tr>
<td>Do not set a datum in the touch probe axis</td>
<td>Q381=0</td>
</tr>
<tr>
<td>No function</td>
<td>Q382=+0</td>
</tr>
<tr>
<td>No function</td>
<td>Q383=+0</td>
</tr>
<tr>
<td>No function</td>
<td>Q384=+0</td>
</tr>
<tr>
<td>Safety clearance in addition to MP6140</td>
<td>Q320=0</td>
</tr>
<tr>
<td>Activate new preset with Cycle 247</td>
<td>Q339=1</td>
</tr>
<tr>
<td>Part program call</td>
<td>Q393=1</td>
</tr>
</tbody>
</table>

4 CYCL DEF 247 DATUM SETTING

6 CALL PGM 35KLZ

7 END PGM CYC416 MM
Touch Probe Cycles: Automatic Workpiece Inspection
16.1 Fundamentals

Overview

The TNC offers twelve cycles for measuring workpieces automatically.

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 REFERENCE PLANE Measuring a coordinate in a selectable axis</td>
<td></td>
<td>Page 402</td>
</tr>
<tr>
<td>1 POLAR DATUM PLANE Measuring a point in a probing direction</td>
<td></td>
<td>Page 403</td>
</tr>
<tr>
<td>420 MEASURE ANGLE Measuring an angle in the working plane</td>
<td></td>
<td>Page 405</td>
</tr>
<tr>
<td>421 MEASURE HOLE Measuring the position and diameter of a hole</td>
<td></td>
<td>Page 408</td>
</tr>
<tr>
<td>422 MEAS. CIRCLE OUTSIDE Measuring the position and diameter of a circular stud</td>
<td></td>
<td>Page 412</td>
</tr>
<tr>
<td>423 MEAS. RECTAN. INSIDE Measuring the position, length and width of a rectangular pocket</td>
<td></td>
<td>Page 416</td>
</tr>
<tr>
<td>424 MEAS. RECTAN. OUTSIDE Measuring the position, length and width of a rectangular stud</td>
<td></td>
<td>Page 420</td>
</tr>
<tr>
<td>425 MEASURE INSIDE WIDTH (2nd soft-key row) Measuring slot width</td>
<td></td>
<td>Page 424</td>
</tr>
<tr>
<td>426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge</td>
<td></td>
<td>Page 427</td>
</tr>
<tr>
<td>427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis</td>
<td></td>
<td>Page 430</td>
</tr>
<tr>
<td>430 MEAS. BOLT HOLE CIRC. (2nd soft-key row) Measuring position and diameter of a bolt hole circle</td>
<td></td>
<td>Page 433</td>
</tr>
<tr>
<td>431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane</td>
<td></td>
<td>Page 437</td>
</tr>
</tbody>
</table>
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 the program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log as a file, the TNC, by default, saves the measuring log as an ASCII file in the directory from which you run the measuring program. As an alternative, you can also send the measuring log directly to a printer or transfer it to a PC via the data interface. To do this, set the print function (in the interface configuration menu) to RS232:\ (see also the User’s Manual under “MOD Functions, Setting Up the Data Interface”).

All measured values listed in the log file are referenced to the datum active during the respective cycle you are running. In addition, the coordinate system may have been rotated in the plane or the plane may have been tilted by using 3D-ROT. In this case, the TNC converts the measuring results to the respective active coordinate system.

Use the HEIDENHAIN data transfer software TNCremo if you wish to output the measuring log via the data interface.
Example: Measuring log for touch probe cycle 421:

**Measuring log for Probing Cycle 421 Hole Measuring**

Date: 30-06-2005  
Time: 6:55:04  
Measuring program: TNC\GEH35712\CHECK1.H

Nominal values:
- Center in reference axis: 50.0000
- Center in minor axis: 65.0000
- Diameter: 12.0000

Given limit values:
- Maximum dimension 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 that are 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:

<table>
<thead>
<tr>
<th>Class of results</th>
<th>Parameter value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measurement results are within tolerance</td>
<td>Q180 = 1</td>
</tr>
<tr>
<td>Rework is required</td>
<td>Q181 = 1</td>
</tr>
<tr>
<td>Scrap</td>
<td>Q182 = 1</td>
</tr>
</tbody>
</table>

The TNC sets the rework or scrap marker as soon as one of the measuring values falls outside of tolerance. To determine which of the measuring results lies outside of tolerance, check the measuring log, or compare the respective measuring results (Q150 to Q160) with their limit values.

In Cycle 427 the TNC assumes that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

The TNC also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.
Tolerance monitoring

For most of the cycles for workpiece inspection you can have the TNC perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the 0 (the default value) in the monitoring parameters.

Tool monitoring

For some cycles for workpiece inspection you can have the TNC perform tool monitoring. The TNC then monitors whether

- The tool radius should be compensated because of the deviations from the nominal value (values in Q16x).
- The deviations from the nominal value (values in Q16x) are greater than the tool breakage tolerance.

Tool compensation

This function works only:

- If the tool table is active.
- If tool monitoring is switched on in the cycle (enter a tool name or Q330 unequal to 0). Select the tool name input by soft key. The TNC no longer displays the right single quotation mark.

If you perform several compensation measurements, the TNC adds the respective measured deviation to the value stored in the tool table.

The TNC always compensates the tool radius in the DR column of the tool table, even if the measured deviation lies within the given tolerance. You can inquire whether re-working is necessary via Parameter Q181 in the NC program (Q181=1: must be reworked).

For Cycle 427:

- If an axis of the active working plane is defined as measuring axis (Q272 = 1 or 2), the TNC compensates the tool radius as described above. From the defined traversing direction (Q267) the TNC determines the direction of compensation.
- If the touch probe axis is defined as measuring axis (Q272 = 3), the TNC compensates the tool length.
**Tool breakage monitoring**

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 protocol file in the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system.
16.2 REF. PLANE (Cycle 0, DIN/ISO: G55)

Cycle run
1. The touch probe moves at rapid traverse (value from MP6150) to the starting position 1 programmed in the cycle.
2. Then the touch probe approaches the workpiece at the feed rate assigned in MP6120. The probing direction is to be defined in the cycle.
3. After the TNC has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in the parameters Q115 to Q119. For the values in these parameters the TNC does not account for the stylus length and radius.

Please note while programming:

Danger of collision!
Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

Cycle parameters

- **Parameter number for result:** Enter the number of the Q parameter to which you want to assign the coordinate. Input range 0 to 1999
- **Probing axis/Probing direction:** Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Confirm your entry with the ENT key. Input range: All NC axes
- **Nominal position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- To conclude the input, press the ENT key.

Example: 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 REFERENCE 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 MP6150) to the starting position programmed in the cycle.

2 Then the touch probe approaches the workpiece at the feed rate assigned in MP6120. During probing the TNC moves simultaneously in 2 axes (depending on the probing angle). The scanning 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
16.3 POLAR REFERENCE PLANE (Cycle 1)

Cycle parameters

▶ **Probing axis:** Enter the probing axis with the axis selection keys or ASCII keyboard. Confirm your entry with the ENT key. Input range: \(X, Y\) or \(Z\)

▶ **Probing angle:** Angle, measured from the probing axis, at which the touch probe is to move. Input range -180.0000 to 180.0000

▶ **Nominal position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999

▶ To conclude the input, press the ENT key.

**Example: NC blocks**

67 TCH PROBE 1.0 POLAR REFERENCE PLANE

68 TCH PROBE 1.1 X ANGLE: +30

69 TCH PROBE 1.2 X+5 Y+0 Z-5
16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

Cycle run

Touch Probe Cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.

1 Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the programmed probe starting point 1 at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

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 saves the measured angle in the following Q parameter:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q150</td>
<td>The measured angle is referenced to the reference axis of the machining plane.</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If touch probe axis = measuring axis, set Q263 equal to Q265 if the angle about the A axis is to be measured; set Q263 not equal to Q265 if the angle is to be measured about the B axis.
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
 Traverse direction 1 Q267: Direction in which the probe is to approach the workpiece:
-1: Negative traverse direction
+1: Positive traverse direction

 Measuring height in the touch probe axis Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

 Setup clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

 Clearance height Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF

 Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

 Measuring log Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR420.TXT in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

Example: NC blocks

```
5 TCH PROBE 420 MEASURE ANGLE
Q263=+10 ;1ST POINT 1ST AXIS
Q264=+10 ;1ST POINT 2ND AXIS
Q265=+15 ;2ND POINT 1ST AXIS
Q266=+95 ;2ND POINT 2ND AXIS
Q272=1 ;MEASURING AXIS
Q267=-1 ;TRAVERSE DIRECTION
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SETUP 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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q153</td>
<td>Actual value of diameter</td>
</tr>
<tr>
<td>Q161</td>
<td>Deviation at center of reference axis</td>
</tr>
<tr>
<td>Q162</td>
<td>Deviation at center of minor axis</td>
</tr>
<tr>
<td>Q163</td>
<td>Deviation from diameter</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the hole dimensions. Minimum input value: 5°.
Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q274 (absolute value): Center of the hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Nominal diameter** Q262: Enter the diameter of the hole. Input range 0 to 99999.9999

- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000

- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000
Measuring height in the touch probe axis
Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

Setup clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

Clearance height Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF

Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Maximum limit of size for hole Q275: Maximum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999

Minimum limit of size for hole Q276: Minimum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999

Tolerance for center 1st axis Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999

Tolerance for center 2nd axis Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
Measuring log Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR421.TXT in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

PGM stop if tolerance error Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
0: Do not interrupt program run, no error message
1: Interrupt program run, output an error message

Tool for monitoring Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
0: Monitoring not active
>0: Tool number in the tool table TOOL.T

No. of measuring points (4/3) Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
4: Use 4 measuring points (standard setting)
3: Use 3 measuring points

Type of traverse? Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between the measuring points if “traverse to clearance height” (Q301=1) is active.
0: Move between operations on a straight line
1: Move between operations on the pitch circle

Example: 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
Q20=0 ;SETUP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=1 ;MOVE TO CLEARANCE
Q275=75.12 ;MAX. LIMIT
Q276=74.95 ;MIN. LIMIT
Q279=0.1 ;TOLERANCE 1ST CENTER
Q280=0.1 ;TOLERANCE 2ND CENTER
Q281=1 ;MEASURING LOG
Q309=0 ;PGM STOP IF ERROR
Q330= ;TOOL
Q423=4 ;NO. OF MEAS. POINTS
Q365=1 ;TYPE OF TRAVERSE
```
16.6 MEAS. CIRCLE 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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q153</td>
<td>Actual value of diameter</td>
</tr>
<tr>
<td>Q161</td>
<td>Deviation at center of reference axis</td>
</tr>
<tr>
<td>Q162</td>
<td>Deviation at center of minor axis</td>
</tr>
<tr>
<td>Q163</td>
<td>Deviation from diameter</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the dimensions of the stud. Minimum input value: 5°
Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the diameter of the stud. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000
414

Touch Probe Cycles: Automatic Workpiece Inspection

16.6 MEAS. CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422)

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternatively **PREDEF**

- **Maximum limit of size for stud** Q277: Maximum permissible diameter for the stud. Input range 0 to 99999.9999

- **Minimum limit of size for the stud** Q278: Minimum permissible diameter for the stud. Input range 0 to 99999.9999

- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999

- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
Measuring log Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR422.TXT in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

PGM stop if tolerance error Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
0: Do not interrupt program run, no error message
1: Interrupt program run, output an error message

Tool number for monitoring Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400): Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
0: Monitoring not active
>0: Tool number in the tool table TOOLT

No. of measuring points (4/3) Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
4: Use 4 measuring points (standard setting)
3: Use 3 measuring points

Type of traverse? Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between the measuring points if “traverse to clearance height” (Q301=1) is active.
0: Move between operations on a straight line
1: Move between operations on the pitch circle

Example: NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>TCH PROBE 422 MEAS. CIRCLE OUTSIDE</td>
<td>Q273=+50 ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td></td>
<td>Q274=+50 ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td></td>
<td>Q262=75 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td></td>
<td>Q325=+90 ;STARTING ANGLE</td>
</tr>
<tr>
<td></td>
<td>Q247=+30 ;STEPPING ANGLE</td>
</tr>
<tr>
<td></td>
<td>Q261=-5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td></td>
<td>Q320=0 ;SETUP CLEARANCE</td>
</tr>
<tr>
<td></td>
<td>Q260=+10 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td></td>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td></td>
<td>Q275=35.1 ;MAX. LIMIT</td>
</tr>
<tr>
<td></td>
<td>Q276=34.9 ;MIN. LIMIT</td>
</tr>
<tr>
<td></td>
<td>Q279=0.05 ;TOLERANCE 1ST CENTER</td>
</tr>
<tr>
<td></td>
<td>Q280=0.05 ;TOLERANCE 2ND CENTER</td>
</tr>
<tr>
<td></td>
<td>Q281=1 ;MEASURING LOG</td>
</tr>
<tr>
<td></td>
<td>Q309=0 ;PGM STOP IF ERROR</td>
</tr>
<tr>
<td></td>
<td>Q330= ;TOOL</td>
</tr>
<tr>
<td></td>
<td>Q423=4 ;NO. OF MEAS. POINTS</td>
</tr>
<tr>
<td></td>
<td>Q365=1 ;TYPE OF TRAVERSE</td>
</tr>
</tbody>
</table>
16.7 MEAS. RECTAN. 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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

3. Then the touch probe moves either paraxially at the measuring height or linearly at the 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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q154</td>
<td>Actual value of length in the reference axis</td>
</tr>
<tr>
<td>Q155</td>
<td>Actual value of length in the minor axis</td>
</tr>
<tr>
<td>Q161</td>
<td>Deviation at center of reference axis</td>
</tr>
<tr>
<td>Q162</td>
<td>Deviation at center of minor axis</td>
</tr>
<tr>
<td>Q164</td>
<td>Deviation of side length in reference axis</td>
</tr>
<tr>
<td>Q165</td>
<td>Deviation of side length in minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **First side length** Q282: Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q283: Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
  Alternatively **PREDEF**

- **Max. size limit 1st side length** Q284: Maximum permissible length of the pocket. Input range 0 to 99999.9999

- **Min. size limit 1st side length** Q285: Minimum permissible length of the pocket. Input range 0 to 99999.9999

- **Max. size limit 2nd side length** Q286: Maximum permissible width of the pocket. Input range 0 to 99999.9999

- **Min. size limit 2nd side length** Q287: Minimum permissible width of the pocket. Input range 0 to 99999.9999

- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999

- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
**Measuring log** Q281: Definition of whether the TNC is to create a measuring log:

0: No measuring log

1: Generate measuring log: with the standard setting the TNC saves the log file **TCHPR423.TXT** in the directory in which your measuring program is also stored.

2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

**PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:

0: Do not interrupt program run, no error message

1: Interrupt program run, output an error message

**Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters

0: Monitoring not active

>0: Tool number in the tool table TOOL.T

---

**Example: NC blocks**

<table>
<thead>
<tr>
<th>NC Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q273=+50 ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q274=+50 ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q282=80 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q283=60 ;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+10 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=1 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q284=0 ;MAX. LIMIT 1ST SIDE</td>
</tr>
<tr>
<td>Q285=0 ;MIN. LIMIT 1ST SIDE</td>
</tr>
<tr>
<td>Q286=0 ;MAX. LIMIT 2ND SIDE</td>
</tr>
<tr>
<td>Q287=0 ;MIN. LIMIT 2ND SIDE</td>
</tr>
<tr>
<td>Q279=0 ;TOLERANCE 1ST CENTER</td>
</tr>
<tr>
<td>Q280=0 ;TOLERANCE 2ND CENTER</td>
</tr>
<tr>
<td>Q281=1 ;MEASURING LOG</td>
</tr>
<tr>
<td>Q309=0 ;PGM STOP IF ERROR</td>
</tr>
<tr>
<td>Q330= ;TOOL</td>
</tr>
</tbody>
</table>
Cycle run

Touch Probe Cycle 424 finds the center, length and width of a rectangular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).

3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point and probes the second touch point.

4. The TNC positions the probe to starting point and then to starting point 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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q154</td>
<td>Actual value of length in the reference axis</td>
</tr>
<tr>
<td>Q155</td>
<td>Actual value of length in the minor axis</td>
</tr>
<tr>
<td>Q161</td>
<td>Deviation at center of reference axis</td>
</tr>
<tr>
<td>Q162</td>
<td>Deviation at center of minor axis</td>
</tr>
<tr>
<td>Q164</td>
<td>Deviation of side length in reference axis</td>
</tr>
<tr>
<td>Q165</td>
<td>Deviation of side length in minor axis</td>
</tr>
</tbody>
</table>
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **First side length** Q282: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q283: Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
Setup clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

Clearance height Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively PREDEF

Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Max. size limit 1st side length Q284: Maximum permissible length of the stud. Input range 0 to 99999.9999

Min. size limit 1st side length Q285: Minimum permissible length of the stud. Input range 0 to 99999.9999

Max. size limit 2nd side length Q286: Maximum permissible width of the stud. Input range 0 to 99999.9999

Min. size limit 2nd side length Q287: Minimum permissible width of the stud. Input range 0 to 99999.9999

Tolerance for center 1st axis Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999

Tolerance for center 2nd axis Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
Measuring log Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR424.TXT in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

PGM stop if tolerance error Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
0: Do not interrupt program run, no error message
1: Interrupt program run, output an error message

Tool number for monitoring Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters:
0: Monitoring not active
>0: Tool number in the tool table TOOL.T

Example: NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 424 MEAS. RECTAN. OUTS.</td>
<td></td>
</tr>
<tr>
<td>Q273=+50</td>
<td>CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q274=+50</td>
<td>CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q282=75</td>
<td>FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q283=35</td>
<td>2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q261=-5</td>
<td>MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0</td>
<td>SETUP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20</td>
<td>CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q284=75.1</td>
<td>MAX. LIMIT 1ST SIDE</td>
</tr>
<tr>
<td>Q285=74.9</td>
<td>MIN. LIMIT 1ST SIDE</td>
</tr>
<tr>
<td>Q286=35</td>
<td>MAX. LIMIT 2ND SIDE</td>
</tr>
<tr>
<td>Q287=34.95</td>
<td>MIN. LIMIT 2ND SIDE</td>
</tr>
<tr>
<td>Q279=0.1</td>
<td>TOLERANCE 1ST CENTER</td>
</tr>
<tr>
<td>Q280=0.1</td>
<td>TOLERANCE 2ND CENTER</td>
</tr>
<tr>
<td>Q281=1</td>
<td>MEASURING LOG</td>
</tr>
<tr>
<td>Q309=0</td>
<td>PGM STOP IF ERROR</td>
</tr>
<tr>
<td>Q330=</td>
<td>TOOL</td>
</tr>
</tbody>
</table>
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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 in the following Q parameters:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q156</td>
<td>Actual value of measured length</td>
</tr>
<tr>
<td>Q157</td>
<td>Actual value of the centerline</td>
</tr>
<tr>
<td>Q166</td>
<td>Deviation of the measured length</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
Cycle parameters

- **Starting point in 1st axis** Q328 (absolute): Starting point for probing in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 2nd axis** Q329 (absolute): Starting point for probing in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Offset for 2nd measurement** Q310 (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the TNC does not offset the touch probe. Input range -99999.9999 to 99999.9999

- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis

- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

- **Nominal length** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999

- **Maximum dimension** Q288: Maximum permissible length. Input range 0 to 99999.9999

- **Minimum dimension** Q289: Minimum permissible length. Input range 0 to 99999.9999
Measuring log Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR425.TXT in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

PGM stop if tolerance error Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
0: Do not interrupt program run, no error message
1: Interrupt program run, output an error message

Tool number for monitoring Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400): Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
0: Monitoring not active
>0: Tool number in the tool table TOOL.T

Setup clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF

Traversing to clearance height Q301: Definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Alternatively PREDEF

Example: 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 MEASUREMENT
Q272=1 ;MEASURING AXIS
Q261=-5 ;MEASURING HEIGHT
Q260=+10 ;CLEARANCE HEIGHT
Q311=25 ;NOMINAL LENGTH
Q288=25.05;MAX. LIMIT
Q289=25 ;MIN. LIMIT
Q281=1 ;MEASURING LOG
Q309=0 ;PGM STOP IF ERROR
Q330= ;TOOL
Q320=0 ;SETUP CLEARANCE
Q301=0 ;MOVE TO CLEARANCE
```
16.10 MEASURE RIDGE WIDTH
(Cycle 426, DIN/ISO: G426)

Cycle run

Touch Probe Cycle 426 measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). 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 in the following Q parameters:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q156</td>
<td>Actual value of measured length</td>
</tr>
<tr>
<td>Q157</td>
<td>Actual value of the centerline</td>
</tr>
<tr>
<td>Q166</td>
<td>Deviation of the measured length</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Ensure that the first measurement is always carried out in the negative direction of the selected measuring axis. Define Q263 and Q264 correspondingly.
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**
- **Nominal length** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999
- **Maximum dimension** Q288: Maximum permissible length. Input range 0 to 99999.9999
- **Minimum dimension** Q289: Minimum permissible length. Input range 0 to 99999.9999
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR426.TXT in the directory in which your measuring program is also stored.
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message

- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

### Example: NC blocks

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TCH PROBE 426 MEASURE RIDGE WIDTH</td>
<td></td>
</tr>
<tr>
<td>Q263=+50 ;1ST POINT 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td>Q264=+25 ;1ST POINT 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td>Q265=+50 ;2ND POINT 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td>Q266=+85 ;2ND POINT 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td>Q272=2 ;MEASURING AXIS</td>
<td></td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
<td></td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td>Q272=45 ;NOMINAL LENGTH</td>
<td></td>
</tr>
<tr>
<td>Q288=45 ;MAX. LIMIT</td>
<td></td>
</tr>
<tr>
<td>Q289=44.95;MIN. LIMIT</td>
<td></td>
</tr>
<tr>
<td>Q281=1 ;MEASURING LOG</td>
<td></td>
</tr>
<tr>
<td>Q309=0 ;PGM STOP IF ERROR</td>
<td></td>
</tr>
<tr>
<td>Q330= TOOL</td>
<td></td>
</tr>
</tbody>
</table>

---

**16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)**
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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe to the probe starting point 1 at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite 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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q160</td>
<td>Measured coordinate</td>
</tr>
</tbody>
</table>

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
Cycle parameters

**1st meas. point 1st axis** Q263 (absolute):
Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

**1st meas. point 2nd axis** Q264 (absolute):
Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

**Measuring height in the touch probe axis** Q261 (absolute):
Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999

**Setup clearance** Q320 (incremental):
Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

**Measuring axis (1..3: 1=reference axis)** Q272:
Axis in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis
3: Touch probe axis = measuring axis

**Traversal direction 1** Q267:
Direction in which the probe is to approach the workpiece:
-1: Negative traverse direction
+1: Positive traverse direction

**Clearance height** Q260 (absolute):
Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**
**Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
- 0: No measuring log
- 1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR427.TXT in the directory in which your measuring program is also stored.
- 2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

**Maximum limit of size** Q288: Maximum permissible measured value. Input range 0 to 99999.999

**Minimum limit of size** Q289: Minimum permissible measured value. Input range 0 to 99999.999

**PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
- 0: Do not interrupt program run, no error message
- 1: Interrupt program run, output an error message

**Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring” on page 400). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters:
- 0: Monitoring not active
- >0: Tool number in the tool table TOOL.T

---

**Example: NC blocks**

```plaintext
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
Q288=5.1 ;MAX. LIMIT
Q289=4.95 ;MIN. LIMIT
Q309=0 ;PGM STOP IF ERROR
Q330= ;TOOL
```
16.12 MEAS. BOLT HOLE CIRC.  
(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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) to the point entered as 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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q151</td>
<td>Actual value of center in reference axis</td>
</tr>
<tr>
<td>Q152</td>
<td>Actual value of center in minor axis</td>
</tr>
<tr>
<td>Q153</td>
<td>Actual value of bolt hole circle diameter</td>
</tr>
<tr>
<td>Q161</td>
<td>Deviation at center of reference axis</td>
</tr>
<tr>
<td>Q162</td>
<td>Deviation at center of minor axis</td>
</tr>
<tr>
<td>Q163</td>
<td>Deviation of bolt hole circle diameter</td>
</tr>
</tbody>
</table>

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, no automatic tool compensation.
Cycle parameters

- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the bolt hole circle diameter. Input range 0 to 99999.9999
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
• **Measuring height in the touch probe axis**
  Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999.

• **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**.

• **Maximum limit of size** Q288: Maximum permissible diameter of bolt hole circle. Input range 0 to 99999.9999.

• **Minimum limit of size** Q289: Minimum permissible diameter of bolt hole circle. Input range 0 to 99999.9999.

• **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999.

• **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999.
Measuring log Q281: Definition of whether the TNC is to create a measuring log:

0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR430.TXT in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

PGM stop if tolerance error Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:

0: Do not interrupt program run, no error message
1: Interrupt program run, output an error message

Tool number for monitoring Q330: Definition of whether the TNC is to monitor for tool breakage (see “Tool monitoring” on page 400): Input range: 0 to 32767.9, alternatively tool name with max. 16 characters.

0: Monitoring not active
>0: Tool number in the tool table TOOL.T

Example: 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 ;MAX. LIMIT
Q289=79.9 ;MIN. LIMIT
Q279=0.15 ;TOLERANCE 1ST CENTER
Q280=0.15 ;TOLERANCE 2ND CENTER
Q281=1 ;MEASURING LOG
Q309=0 ;PGM STOP IF ERROR
Q330= ;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 (see “Executing touch probe cycles” on page 318), the TNC positions the touch probe at rapid traverse (value from MP6150) to the programmed starting point and measures the first touch 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 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 and measures the actual value of the third touch point.

4. Finally the TNC returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q158</td>
<td>Projection angle of the A axis</td>
</tr>
<tr>
<td>Q159</td>
<td>Projection angle of the B axis</td>
</tr>
<tr>
<td>Q170</td>
<td>Spatial angle A</td>
</tr>
<tr>
<td>Q171</td>
<td>Spatial angle B</td>
</tr>
<tr>
<td>Q172</td>
<td>Spatial angle C</td>
</tr>
<tr>
<td>Q173 to Q175</td>
<td>Measured values in the touch probe axis</td>
</tr>
<tr>
<td></td>
<td>(first to third measurement)</td>
</tr>
</tbody>
</table>
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.

If you run the cycle while a tilted working plane is active, the spatial angle is measured with respect to the tilted coordinate system. In this case, use the measured spatial angle with `PLANE RELATIV`. 
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **1st meas. point 3rd axis** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999

- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **2nd meas. point 3rd axis** Q295 (absolute): Coordinate of the second touch point in the touch probe axis. Input range -99999.9999 to 99999.9999

- **3rd meas. point 1st axis** Q266 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd meas. point 2nd axis** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd meas. point 3rd axis** Q298 (absolute): Coordinate of the third touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
**Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

**Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999, alternatively **PREDEF**

**Measuring log** Q281: Definition of whether the TNC is to create a measuring log:

0: No measuring log

1: Generate measuring log: with the standard setting the TNC saves the log file TCHPR431.TXT in the directory in which your measuring program is also stored.

2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

---

**Example: NC blocks**

<table>
<thead>
<tr>
<th>Step</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>TCH PROBE 431 MEASURE PLANE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q263=+20 ;1ST POINT 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q264=+20 ;1ST POINT 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q294=+10 ;1ST POINT 3RD AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q265=+90 ;2ND POINT 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q266=+25 ;2ND POINT 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q295=+15 ;2ND POINT 3RD AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q296=+50 ;3RD POINT 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q297=+80 ;3RD POINT 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q298=+20 ;3RD POINT 3RD AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q320=0 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q260=+5 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q281=1 ;MEASURING LOG</td>
<td></td>
</tr>
</tbody>
</table>
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  
2 L Z+100 R0 FMAX  
3 FN 0: Q1 = +81  
4 FN 0: Q2 = +61  
5 CALL LBL 1  
6 L Z+100 R0 FMAX  
7 TOOL CALL 99 Z  
8 TCH PROBE 424 MEAS. RECTAN. OUTS.  
   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 ;SETUP CLEARANCE  
   Q260=+30 ;CLEARANCE HEIGHT  
   Q301=0 ;MOVE TO CLEARANCE  
   Q284=0 ;MAX. LIMIT 1ST SIDE 
```

Prepare tool call
Retract the tool
Pocket length in X (roughing dimension)
Pocket length in Y (roughing dimension)
Call subprogram for machining
Retract the tool, change the tool
Call the touch probe
Measure the rough-milled rectangle
Nominal length in X (final dimension)
Nominal length in Y (final dimension)
Input values for tolerance checking not required
### Touch Probe Cycles: Automatic Workpiece Inspection

#### 16.14 Programming Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>CYCL DEF 213 STUD FINISHING</td>
<td>Subprogram with fixed cycle for rectangular studs</td>
</tr>
<tr>
<td>17</td>
<td>CYCL CALL M3</td>
<td>Cycle call</td>
</tr>
<tr>
<td>18</td>
<td>LBL 0</td>
<td>End of subprogram</td>
</tr>
<tr>
<td>19</td>
<td>END PGM BEAMS MM</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Q285=0 ;MIN. LIMIT 1ST SIDE</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Q286=0 ;MAX. LIMIT 2ND SIDE</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Q287=0 ;MIN. LIMIT 2ND SIDE</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Q279=0 ;TOLERANCE 1ST CENTER</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>Q280=0 ;TOLERANCE 2ND CENTER</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Q281=0 ;MEASURING LOG</td>
<td>No measuring log transmission</td>
</tr>
<tr>
<td>7</td>
<td>Q309=0 ;PGM STOP IF ERROR</td>
<td>Do not output an error message</td>
</tr>
<tr>
<td>8</td>
<td>Q330=0 ;TOOL NUMBER</td>
<td>No tool monitoring</td>
</tr>
<tr>
<td>9</td>
<td>9 FN 2: Q1 = +Q1 - +Q164</td>
<td>Calculate length in X including the measured deviation</td>
</tr>
<tr>
<td>10</td>
<td>10 FN 2: Q2 = +Q2 - +Q165</td>
<td>Calculate length in Y including the measured deviation</td>
</tr>
<tr>
<td>11</td>
<td>11 L Z+100 R0 FMAX</td>
<td>Retract the touch probe, change the tool</td>
</tr>
<tr>
<td>12</td>
<td>12 TOOL CALL 1 Z S5000</td>
<td>Tool call for finishing</td>
</tr>
<tr>
<td>13</td>
<td>13 CALL LBL 1</td>
<td>Call subprogram for machining</td>
</tr>
<tr>
<td>14</td>
<td>14 L Z+100 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>15</td>
<td>15 LBL 1</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>Q200=20 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>Q201=-10 ;DEPTH</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>Q206=150 ;FEED RATE FOR PLUNGING</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>Q202=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>Q203=+10 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>Q204=20 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>Q216=+50 ;CENTER IN 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>Q217=+50 ;CENTER IN 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>Q218=Q1 ;FIRST SIDE LENGTH</td>
<td>Length in X variable for roughing and finishing</td>
</tr>
<tr>
<td>26</td>
<td>Q219=Q2 ;2ND SIDE LENGTH</td>
<td>Length in Y variable for roughing and finishing</td>
</tr>
<tr>
<td>27</td>
<td>Q220=0 ;CORNER RADIUS</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>Q221=0 ;ALLOWANCE IN 1ST AX</td>
<td></td>
</tr>
</tbody>
</table>
Example: Measuring a rectangular pocket and recording the results

```
0 BEGIN PGM BSMEAS MM
1 TOOL CALL 1 Z
2 L Z+100 R0 FMAX
3 TCH PROBE 423 MEAS. RECTAN. INSIDE
   Q273=+50 ;CENTER IN 1ST AXIS
   Q274=+40 ;CENTER IN 2ND AXIS
   Q282=90 ;FIRST SIDE LENGTH
   Q283=70 ;2ND SIDE LENGTH
   Q261=-5 ;MEASURING HEIGHT
   Q320=0 ;SET-UP CLEARANCE
   Q260=+20 ;CLEARANCE HEIGHT
   Q301=0 ;MOVE TO CLEARANCE
```
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q284=90.15; MAX. LIMIT 1ST SIDE</td>
<td>Maximum limit in X</td>
</tr>
<tr>
<td>Q285=89.95; MIN. LIMIT 1ST SIDE</td>
<td>Minimum limit in X</td>
</tr>
<tr>
<td>Q286=70.1; MAX. LIMIT 2ND SIDE</td>
<td>Maximum limit in Y</td>
</tr>
<tr>
<td>Q287=69.9; MIN. LIMIT 2ND SIDE</td>
<td>Minimum limit in Y</td>
</tr>
<tr>
<td>Q279=0.15; TOLERANCE 1ST CENTER</td>
<td>Permissible position deviation in X</td>
</tr>
<tr>
<td>Q280=0.1; TOLERANCE 2ND CENTER</td>
<td>Permissible position deviation in Y</td>
</tr>
<tr>
<td>Q281=1; MEASURING LOG</td>
<td>Save measuring log to a file</td>
</tr>
<tr>
<td>Q309=0; PGM STOP IF ERROR</td>
<td>Do not display an error message in case of a tolerance violation</td>
</tr>
<tr>
<td>Q330=0; TOOL NUMBER</td>
<td>No tool monitoring</td>
</tr>
<tr>
<td>4 L Z+100 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
</tbody>
</table>

触探循环：自动工件检测
Touch Probe Cycles: Special Functions
17.1 Basics

Overview

The TNC provides seven cycles for the following special purposes:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 CALIBRATE TS Radius calibration of the touch trigger probe</td>
<td>2</td>
<td>Page 447</td>
</tr>
<tr>
<td>9 CALIBRATE TS LENGTH Length calibration of the touch trigger probe</td>
<td>9</td>
<td>Page 448</td>
</tr>
<tr>
<td>3 MEASURING Cycle for defining OEM cycles</td>
<td>3</td>
<td>Page 449</td>
</tr>
<tr>
<td>4 MEASURING IN 3-D Measuring cycle for 3-D probing for defining OEM cycles</td>
<td>4</td>
<td>Page 451</td>
</tr>
<tr>
<td>440 MEASURE AXIS SHIFT</td>
<td>440</td>
<td>Page 453</td>
</tr>
<tr>
<td>441 FAST PROBING</td>
<td>441</td>
<td>Page 456</td>
</tr>
<tr>
<td>460 CALIBRATE TS Radius and length calibration on a calibration sphere</td>
<td>460</td>
<td>Page 458</td>
</tr>
</tbody>
</table>
17.2 CALIBRATE TS (Cycle 2)

Cycle run

Touch probe cycle 2 automatically calibrates a touch trigger probe using a ring gauge or a precision stud as calibration standard.

1. The touch probe moves at rapid traverse (value from MP6150) to the clearance height (but only if the current position is below the clearance height).
2. Then the TNC positions the touch probe in the working plane to the center of the ring gauge (calibration from inside) or in its proximity (calibration from outside).
3. The touch probe then moves to the measuring depth (result of machine parameters 618x.2 and 6185.x) and probes the ring gauge successively in X+, Y+, X- and Y-.
4. Finally, the TNC moves the touch probe to the clearance height and writes the effective radius of the ball tip to the calibration data.

Please note while programming:

Before you begin calibrating, you must define in Machine Parameters 6180.0 to 6180.2 the center of the calibrating workpiece in the working space of the machine (REF coordinates).

If you are working with several traverse ranges you can save a separate set of coordinates for the center of each calibrating workpiece (MP6181.1 to 6181.2 and MP6182.1 to 6182.2).

Example: NC blocks

```
5 TCH PROBE 2.0 CALIBRATE TS
6 TCH PROBE
2.1 HEIGHT: +50 R +25.003 DIRECTION: 0
```
17.3 CALIBRATE TS LENGTH
(Cycle 9)

Cycle run

Touch probe cycle 9 automatically calibrates the length of a touch trigger probe at a point that you determine.

1 Pre-position the touch probe so that the coordinate defined in the cycle can be accessed without collision.
2 The TNC moves the touch probe in the direction of the negative tool axis until a trigger signal is released.
3 Finally, the TNC moves the touch probe back to the starting point of the probing process and writes the effective touch probe length into the calibration data.

Cycle parameters

- **Coordinate of datum** (absolute): Exact coordinate of the point that is to be probed. Input range -99999.9999 to 99999.9999
- **Reference system? (0=ACT/1=REF)**: Specify the coordinate system on which the entered datum is to be based:
  0: Entered datum is based on the active workpiece coordinate system (ACT system)
  1: Entered datum is based on the active machine coordinate system (REF system)

Example: NC blocks

```
5 L X-235 Y+356 R0 FMAX
6 TCH PROBE 9.0 CALIBRATE TS LENGTH
7 TCH PROBE 9.1 DATUM +50 REFERENCE SYSTEM 0
```
17.4 MEASURING (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 path DIST 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 machine parameters 6130 (maximum traverse to touch point) and 6120 (probing feed rate), which are effective in other measuring cycles, do not apply in touch probe cycle 3.

Remember that the TNC always writes to 4 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: SYWRITE ID 990 NR 6 you can set whether the cycle runs through the probe input X12 or X13.
Cycle parameters

- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters. Input range 0 to 1999

- **Probing axis**: Enter the axis in whose direction the probe is to move and confirm with the ENT key. Input range: X, Y or Z

- **Probing angle**: Angle, measured from the defined probing axis in which the touch probe is to move. Confirm with ENT. Input range -180.0000 to 180.0000

- **Maximum measuring path**: Enter the maximum distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range -99999.9999 to 99999.9999

- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000

- **Maximum retraction path**: 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)**: Specify whether the probing direction and the result of measurement are to be referenced to the actual coordinate system (ACT, can be shifted or rotated), or to the machine coordinate system (REF):
  - 0: Probe in the current system and save measurement result in the ACT system
  - 1: Probe in the machine-based REF system and save measurement result in 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 you select mode 1, the TNC saves the value 2.0 in the 4th result parameter and continues the cycle.
  - 0: Issue error message
  - 1: Do not issue error message

---

**Example: NC blocks**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>TCH PROBE 3.0 MEASURING</td>
</tr>
<tr>
<td>5</td>
<td>TCH PROBE 3.1 Q1</td>
</tr>
<tr>
<td>6</td>
<td>TCH PROBE 3.2 X ANGLE: +15</td>
</tr>
<tr>
<td>7</td>
<td>TCH PROBE 3.3 DIST +10 F100 MB1 REFERENCE SYSTEM:0</td>
</tr>
<tr>
<td>8</td>
<td>TCH PROBE 3.4 ERRORMODE1</td>
</tr>
</tbody>
</table>
17.5 MEASURING IN 3-D (Cycle 4, FCL 3 function)

Cycle run

Cycle 4 is an auxiliary cycle that you can only use in conjunction with external software! The TNC does not provide any cycle with which you can calibrate the touch probe:

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 path and feed rate directly. Also, the touch probe retracts by a definable value after determining the measured value.

1. The touch probe 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 touch probe stops. The TNC saves the X, Y, Z coordinates of the probe-tip center (without calculation of the calibration data) in three successive Q parameters. You define the number of the first 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 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 4 successive Q parameters. If the TNC could not determine a valid touch point, the fourth result parameter will have the value –1.

The TNC saves the measured values without calculating the calibration data of the touch probe.

With function FN17: SYSWRITE ID 990 NR 6 you can set whether the cycle runs through the probe input X12 or X13.
17.5 MEASURING IN 3-D (Cycle 4, FCL 3 function)

Cycle parameters

- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first coordinate (X). Input range 0 to 1999

- **Relative measuring path in X**: X component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999

- **Relative measuring path in Y**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999

- **Relative measuring path in Z**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999

- **Maximum measuring path**: Enter the maximum distance from the starting point by which the touch probe may move along the direction vector. Input range: -99999.9999 to 99999.9999

- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000

- **Maximum retraction path**: 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 measurement is to be saved in the actual coordinate system (ACT, can be shifted or rotated), or with respect to the machine coordinate system (REF).
  - **0**: Save the measurement result in the ACT system
  - **1**: Save the measurement result in the REF system

**Example: NC blocks**

```
5 TCH PROBE 4.0 MEASURING IN 3-D
6 TCH PROBE 4.1 Q1
7 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
8 TCH PROBE
4.3 DIST +45 F100 MB50 REFERENCE SYSTEM:0
```
17.6 MEASURE AXIS SHIFT
(Touch Probe Cycle 440, DIN/ISO: G440)

Cycle run

Touch probe cycle 440 measures the axis shifts of the machine. Make sure that the cylindrical calibrating tool used in connection with the TT 130 has the correct dimensions.

1 The TNC positions the calibrating tool at rapid traverse (value from MP6550) and following the positioning logic (refer to chapter 1.2) in the vicinity of the TT.

2 At first the TNC makes a measurement in the touch probe axis. The calibrating tool is offset by the value you have defined in the tool table TOOL.T under TT: R-OFFS (standard = tool radius). The TNC always performs the measurement in the touch probe axis.

3 Then the TNC makes the measurement in the working plane. You define via parameter Q364 in which axis and in which direction of the working plane the measurement is to be made.

4 If you make a calibration, the TNC saves the calibration data. Whenever you make a measurement, the TNC compares the measured values to the calibration data and writes the deviations to the following Q parameters:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q185</td>
<td>Deviation from calibration value in X</td>
</tr>
<tr>
<td>Q186</td>
<td>Deviation from calibration value in Y</td>
</tr>
<tr>
<td>Q187</td>
<td>Deviation from calibration value in Z</td>
</tr>
</tbody>
</table>

You can use this value for compensating the deviation through an incremental datum shift (Cycle 7).

5 Finally, the calibrating tool returns to the clearance height.
Please note while programming:

Before running cycle 440 for the first time, you must have calibrated the tool touch probe with tool-touch-probe cycle 30.

Ensure that the tool data of the calibrating tool has been entered in the tool table TOOL.T.

Before running the cycle, you must activate the calibrating tool with TOOL CALL.

Ensure that the TT tool touch probe is connected to input X13 of the logic unit and is ready to function (Machine Parameter 65xx).

Before you perform a measurement, you must have made at least one calibration, otherwise the TNC will output an error message. If you are working with several traverse ranges, you have to make a calibration for each of them.

The TNC calculates incorrect values if the probing directions for calibrating and measuring do not correspond.

Each time you run cycle 440, the TNC resets the result parameters Q185 to Q187.

If you want to set a limit for the axis shift in the machine axes, enter the desired limits in the tool table TOOL.T under LTOL for the spindle axis and under RTOL for the working plane. If the limits are exceeded, the TNC outputs a corresponding error message after a control measurement.

After the cycle is completed, the TNC restores the spindle settings that were active before the cycle (M3/M4).
Cycle parameters

- **Operation**: 0=calibr., 1=measure? Q363: Specify whether you want to calibrate or make a verification measurement:
  0: Calibrate
  1: Measure

- **Probing directions** Q364: Definition of probing direction(s) in the working plane:
  0: Measuring only in the positive direction of the reference axis
  1: Measuring only in the positive direction of the minor axis
  2: Measuring only in the negative direction of the reference axis
  3: Measuring only in the negative direction of the minor axis
  4: Measuring in the positive directions of the reference axis and the minor axis
  5: Measuring in the positive direction of the reference axis and in the negative direction of the minor axis
  6: Measuring in the negative direction of the reference axis and in the positive direction of the minor axis
  7: Measuring in the negative directions of the reference axis and the minor axis

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and probe contact. Q320 is added to MP6540. Input range 0 to 99999.9999, alternatively PREDEF

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur (referenced to the active datum). Input range -99999.9999 to 99999.9999, alternatively PREDEF

---

Example: NC blocks

```
5 TCH PROBE 440 MEASURE AXIS SHIFT
Q363=1 ;DIRECTION
Q364=0 ;PROBING DIRECTIONS
Q320=2 ;SET-UP CLEARANCE
Q260=+50 ;CLEARANCE HEIGHT
```
17.7 FAST PROBING (Cycle 441, DIN/ISO: G441, FCL 2 Function)

Cycle run

Touch probe cycle 441 allows the global setting of different touch probe parameters (e.g. positioning feed rate) for all subsequently used touch probe cycles. This makes it easy to optimize the programs so that reductions in total machining time are achieved.

Please note while programming:

Before programming, note the following

There are no machine movements contained in Cycle 441. It only sets different probing parameters.

END PGM, M02, M30 resets the global settings of Cycle 441.

You can activate automatic angle tracking (Cycle Parameter Q399) only if Machine Parameter 6165=1. If you change Machine Parameter 6165, you must recalibrate the touch probe.
Cycle parameters

- **Positioning feed rate** Q396: Define the feed rate at which the touch probe is moved to the specified positions. Input range 0 to 99999.9999

- **Positioning feed rate=FMAX (0/1)** Q397: Define whether the touch probe is to move at FMAX (rapid traverse) to the specified positions.
  0: Move at feed rate from Q396
  1: Move at FMAX

- **Angle tracking** Q399: Define whether the TNC is to orient the touch probe before each probing process.
  0: Do not orient
  1: Orient the spindle before each probing process to increase the accuracy

- **Automatic interruption** Q400: Define whether the TNC is to interrupt program run and display the measurement results on the screen after a measuring cycle for automatic workpiece measurement:
  0: Never interrupt the program run, not even if the output of the measurement results on the screen is selected in the respective probing cycle.
  1: Always interrupt program run and display the measurement results on the screen. To continue the program run, press the NC Start button

**Example: NC blocks**

```
5 TCH PROBE 441 FAST PROBING
Q396=3000 ;POSITIONING FEED RATE
Q397=0 ;SELECT FEED RATE
Q399=1 ;ANGLE TRACKING
Q400=1 ;INTERRUPTION
```
17.8 CALIBRATE TS (Cycle 460, DIN/ISO: G460)

Cycle run

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically on an exact calibration sphere. You can do radius calibration alone, or radius and length calibration.

1  Clamp the calibration sphere and check for potential collisions.
2  In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3  The first movement in the cycle is in the negative direction of the touch probe axis.
4  Then the cycle determines the exact center of the sphere in the touch probe axis.

Please note while programming:

Before programming, note the following
Pre-position the touch probe in the program so that it is located approximately above the center of the calibration sphere.
Cycle parameters

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively PREDEF
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
  Alternatively PREDEF
- **No. of probe points in plane (4/3)** Q423: Specify whether the TNC should measure the calibration sphere in the plane with 4 or 3 probing points. 3 probing points increase the measuring speed:
  - 4: Use 4 measuring points (standard setting)
  - 3: Use 3 measuring points
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Calibrate length (0/1)** Q433: Define whether the TNC is to calibrate the touch probe length after radius calibration, as well:
  - 0: Do not calibrate touch probe length
  - 1: Calibrate touch probe length
- **Datum for length** Q434 (absolute): Coordinate of the calibration sphere center. The definition is only required if length calibration is to be carried out. Input range: -99999.9999 to 99999.9999

**Example: NC blocks**

```
  5 TCH PROBE 460 CALIBRATE TS
  Q407=12.5 ;SPHERE RADIUS
  Q320=0 ;SETUP CLEARANCE
  Q301=1 ;MOVE TO CLEARANCE
  Q423=4 ;NO. OF PROBE POINTS
  Q380=+0 ;REFERENCE ANGLE
  Q433=0 ;CALIBRATE LENGTH
  Q434=-2.5 ;DATUM
```
17.8 CALIBRATE TS (Cycle 460, DIN/ISO: G460)
Touch Probe Cycles: Automatic Kinematics Measurement
18.1 Kinematic Measurement with TS Touch Probes (Option KinematicsOpt)

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 figure 1 at right), and the kinematic conditions actually existing on the machine (see figure 2 at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see figure 3 at right). It is therefore necessary for the model to approach reality as closely as possible.

The new TNC function **KinematicsOpt** is an important component that helps you to really fulfill these complex requirements: A 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are in the form of tables or spindle heads. A calibration sphere is fixed at any position on the machine table, and measured with a resolution that you define. During cycle definition you simply define for each rotary axis the area that you want to measure.

From the measured values, the TNC calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematic table.

Overview

The TNC offers cycles that enable you to automatically save, check and optimize the machine kinematics:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Soft key</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>450 SAVE KINEMATICS: Automatically saving and restoring kinematic configurations</td>
<td></td>
<td>Page 464</td>
</tr>
<tr>
<td>451 MEASURE KINEMATICS: Automatically checking or optimizing the machine kinematics</td>
<td></td>
<td>Page 466</td>
</tr>
<tr>
<td>452 PRESET COMPENSATION: Automatically checking or optimizing the machine kinematics</td>
<td></td>
<td>Page 482</td>
</tr>
</tbody>
</table>
18.2 Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt) and 8 (software option1) and FCL3 must be enabled.
- Software option 52 (KinematicsComp) is necessary for compensations of angular positions.
- 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 be measured (by the machine tool builder during commissioning).
- Machine parameter MP6600 must define the tolerance limit starting from which the TNC displays a note when the changes in the kinematic data exceed this limit value (see "KinematicsOpt: Tolerance limit in Optimization mode: MP6600" on page 317).
- Machine parameter MP6601 must define the maximum permissible deviation from the entered cycle parameter by the calibration sphere radius measured in the cycles (see "KinematicsOpt, permissible deviation of the calibration ball radius: MP6601" on page 317).
- The M function number to be used for rotary axis positioning must be entered in machine parameter MP 6602, or -1 if positioning is to be done by the NC. An M function must be specially intended for this application by your machine tool builder.

Please note while programming:

The KinematicsOpt cycles use the global string parameters Q50 to Q599. Please note that they may have changed after execution of these cycles.

If MP 6602 is not equal to -1 you have to position the rotary axes to 0 degrees (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450).
18.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450; Option)

Cycle run

With touch probe cycle 450, you can save the active machine kinematics, restore a previously saved one, or output the current saving status on the screen and in a log file. There are 10 memory spaces available (numbers 0 to 9).

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).

Save mode: In addition to the kinematic configuration, the TNC always saves the code number (freely definable) last entered under MOD. Then you cannot overwrite this memory space unless you enter this code number. If you have saved a kinematic configuration without a code number, the TNC automatically overwrites this memory space during the next saving process!

Restore mode: The TNC can restore saved data only to a matching kinematic configuration.

Restore mode: Note that a change in the kinematics always changes the preset as well. Set the preset again if necessary.
### Cycle parameters

- **Mode (0/1/2) Q410:** Specify whether to save or restore a kinematics configuration:
  - 0: Save active kinematics
  - 1: Restore previously saved kinematics configuration
  - 2: Display the saving status
- **Memory (0…9) Q409:** Number of the memory space to which you want to save the entire kinematics configuration, or the number of the memory space from which you want to restore it. Input range 0 to 9, no function if mode 2 is selected.

### Log function

After running Cycle 450, the TNC creates a measuring log (TCHPR450.TXT) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Save/1=Restore/2=Saving status)?
- Number of the memory space (0 to 9)
- Line number of the kinematics configuration in the kinematic table
- Code number, if you entered one immediately before running Cycle 450.

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 with the current saving status on the screen and in the log, including the number of the memory space, code numbers, kinematics numbers and date of saving

### Example: NC blocks

```
5 TCH PROBE 450 SAVE KINEMATICS
Q410=0 ;MODE
Q409=1 ;MEMORY
```
18.4 MEASURE KINEMATICS
(Cycle 451, DIN/ISO: G451; Option)

Cycle run

The touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

HEIDENHAIN recommends using the calibration spheres KKH 250 (ID number 655 475-01) or KKH 100 (ID number 655 475-02), which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The TNC evaluates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

1. Clamp the calibration sphere and check for potential collisions
2. In the Manual Operation mode, set the datum in the center of the sphere, or if Q431=1 or Q431=3 is defined: In the touch probe axis, manually position the touch probe over the calibration sphere, and in the working plane, over the sphere center.
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:

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q141</td>
<td>Standard deviation measured in the A axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q142</td>
<td>Standard deviation measured in the B axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q143</td>
<td>Standard deviation measured in the C axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q144</td>
<td>Optimized standard deviation in the A axis (–1 if axis was not optimized)</td>
</tr>
<tr>
<td>Q145</td>
<td>Optimized standard deviation in the B axis (–1 if axis was not optimized)</td>
</tr>
<tr>
<td>Q146</td>
<td>Optimized standard deviation in the C axis (–1 if axis was not optimized)</td>
</tr>
</tbody>
</table>
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°. The TNC will issue an error message if the selected start angle, end angle and number of measuring points result in a measuring position of 0°.

Specify the start and end angles to ensure that the same position is not measured twice. As mentioned above, a duplicated point measurement (e.g. measuring positions +90° and -270°) is not advisable, however it does not cause an error message.

- Example: start angle = +90°, end angle = –90°
  - Start angle = +90°
  - End angle = –90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (–90 - +90) / (4-1) = –60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = -30°
  - Measuring point 4 = –90°

- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270 - 90) / (4-1) = +60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°
Machines with Hirth-coupled axes

Danger of collision!

In order to be positioned, the axis must move out of the Hirth 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 manufacturer enabling the TNC to move the rotary axes. The machine manufacturer must have entered the number of the M function in machine parameter $MP6602$ 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 = $3^\circ$

Calculated stepping angle = $\frac{Q412 - Q411}{Q414 - 1}$

Calculated stepping angle = $\frac{90 - 30}{4 - 1} = 120 / 3 = 40$

Measuring position 1 = $Q411 + 0 \times$ stepping angle = $-30^\circ \rightarrow -30^\circ$

Measuring position 2 = $Q411 + 1 \times$ stepping angle = $+10^\circ \rightarrow 9^\circ$

Measuring position 3 = $Q411 + 2 \times$ stepping angle = $+50^\circ \rightarrow 51^\circ$

Measuring position 4 = $Q411 + 3 \times$ stepping angle = $+90^\circ \rightarrow 90^\circ$
**Choice of number of measuring points**

To save time you can make a rough optimization with a small number of measuring points (1-2).

You then make a fine optimization with a medium number of measuring points (recommended value = 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° - 360° at 3 measuring points at 90°, 180° and 270°.

If you want to check the accuracy accordingly, you can enter a higher number of measuring points in the **Check** mode.

> You must not define a measuring point at 0° or 360°. These positions do not provide any metrologically relevant data and lead to an error message!

**Choice of the calibration sphere position on the machine table**

In principle, you can fix the calibration sphere to any accessible position to the machine table and also on chucking equipment or workpieces. The following factors can positively influence the result of measurement:

- **On machines with rotary tables/tilting tables:**
  Clamp the calibration sphere as far as possible away from the center of rotation.

- **Machines with large traverse:**
  Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.
Notes on the accuracy

The geometrical and positioning error of the machine influences 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 error, 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 error, the greater is the dispersion of measured results when you fix the calibration sphere to different positions in the machine coordinate system.

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 using machine parameter MP6165. 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 also “Log function” on page 479).

Backlash measurement is not possible if machine parameter MP6602 is set or if the axis is a Hirth axis.
Please note while programming:

Note that all functions for tilting in the working plane are reset. **M128** or **FUNCTION TCPM** are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it, or you define the input parameter Q431 correspondingly to 1 or 3.

If machine parameter **MP6602** is not equal -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 machine parameter **MP6150**, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate **Q253**, while the probe monitoring is inactive.

If the kinematic data attained in the optimize mode are greater than the permissible limit (**MP6600**), 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 **MP6601**, the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.
**Cycle parameters**

- **Mode (0=Check/1=Measure)** Q406: Specify whether the TNC should check or optimize the active kinematics:
  - 0: Check the active machine kinematics. The TNC measures the kinematics in the 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 the active machine kinematics. The TNC measures the kinematics in the rotary axes you have defined and optimizes the position of the rotary axes of the active kinematics.
  - 2: Optimize the active machine kinematics. The TNC measures the kinematics in the rotary axes you have defined and optimizes the position and compensates the angle of the rotary axes of the active kinematics. The KinematicsComp option must be enabled for Mode 2.

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the series A, then B, then C.
  - Input >0:
    Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. The probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.

**Example: Calibration program**

```
4 TOOL CALL "TASTER" Z
5 TCH PROBE 450 SAVE KINEMATICS
Q410=0 ;MODE
Q409=5 ;MEMORY
6 TCH PROBE 451 MEASURE KINEMATICS
Q406=1 ;MODE
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SETUP 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 ;INICID. ANGLE A AXIS
Q414=0 ;MEAS. POINTS A AXIS
Q415=-90 ;START ANGLE B AXIS
Q416=+90 ;END ANGLE B AXIS
Q417=0 ;INICID. ANGLE B AXIS
Q418=2 ;MEAS. POINTS B AXIS
Q419=-90 ;START ANGLE C AXIS
Q420=+90 ;END ANGLE C AXIS
Q421=0 ;INICID. ANGLE C AXIS
Q422=2 ;MEAS. POINTS C AXIS
Q423=4 ;NO. OF MEAS. POINTS
Q431=1 ;PRESET
Q432=0 ;BACKLASH, ANG. RANGE
```
Feed rate for pre-positioning Q253: Traversing speed of the tool during positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively FMAX, FAUTO, PREDEF.

Reference angle Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000

Start angle A axis Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999

End angle A axis Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999

Angle of incid. A axis Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

Number meas. points A axis Q414: Number of probe measurements with which the TNC is to measure the A axis. If input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

Start angle B axis Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999

End angle B axis Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999

Angle of incid. in B axis Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

Number meas. points B axis Q418: Number of probe measurements with which the TNC is to measure the B axis. If input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. Input range 0 to 12. If input value = 0, the TNC does not measure the respective axis.
- **No. of measuring points (4/3) Q423**: Specify whether the TNC should measure the calibration sphere in the plane with 4 or 3 probing points. 3 probing points increase the measuring speed:
  - 4: Use 4 measuring points (standard setting)
  - 3: Use 3 measuring points
- **Preset (0/1/2/3) Q431**: Specify whether the TNC is to set the active preset (datum) automatically in the center of the sphere:
  - 0: Do not set the preset automatically in the center of the sphere: Set the preset manually before the start of the cycle
  - 1: Set the preset automatically in the center of the sphere before measurement: Preposition the touch probe manually over the calibration sphere before the start of the cycle
  - 2: Set the preset automatically in the center of the sphere after measurement: Set the preset manually before the start of the cycle
  - 3: Set the preset automatically in the center of the sphere before and after measurement: Preposition the touch probe manually over the calibration sphere before the start of the cycle
- **Backlash, angle range Q432**: Here you define the angle value to be used as traverse for the measurement of the rotary axis. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range: -3.0000 to +3.0000

If you have activated "Preset" before measurement (Q431 = 1/3), then move the touch probe to a position above the center of the calibration sphere before the start of the cycle.
Various modes (Q406)

- **Test mode Q406 = 0**
  - The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
  - The TNC records the results of a possible position optimization but does not make any adjustments.

- **Position Optimization mode Q406 = 1**
  - The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
  - During this, the TNC tries to change the position of the rotary 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 position of the rotary axis by means of compensation (Option #52, KinematicsComp).
  - If the TNC was able to optimize the angle, it automatically optimizes the position in another measurement series.

For angle optimization, the machine manufacturer must have adapted the configuration correspondingly. You can ask your machine manufacturer if this is the case, and whether an angle optimization makes sense. Angle optimization can be particularly useful on small, compact machines.

Angle compensation is only possible with Option #52 KinematicsComp.

Example: Angle and position optimization of the rotary axes after automatic datum setting

```plaintext
1 TOOL CALL "TASTER" Z
2 TCH PROBE 451 MEASURE KINEMATICS
Q406=2 ;MODE
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SETUP 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=4 ;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 MEAS. POINTS
Q431=1 ;PRESET
Q432=0 ;BACKLASH, ANG. RANGE
```
Log 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
  - Ascertained backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Evaluation of measuring points
  - Measurement uncertainty of rotary axes
Notes on log data

Error outputs
In the Test mode (Q406=0) the TNC outputs the accuracy that can be attained by optimization and/or the accuracies attained through optimization (Modes 1 and 2).
If the angular position of a rotary axis was calculated, the measured data is also shown in the log.

Dispersion (standard deviation)
In the log, ‘dispersion’, a term from statistics, is used as a measure of accuracy. **Measured dispersion** (measured standard deviation) means that 68.3% of the actually measured spatial errors are within the specified range (+/–). **Optimized dispersion** (optimized standard deviation) means that 68.3% of the spatial errors to be expected after the correction of the kinematics are within the specified range (+/–).

Evaluation of measuring points
The valuation numbers are a measure of the quality of the measuring positions with respect to the changeable transformations of the kinematics model. The higher the valuation number, the greater the benefit from optimization by the TNC.

Because the TNC always needs two transformations for measuring the position of a rotary axis, two valuation numbers per rotary axis are determined. If a complete valuation is missing, the position of a rotary axis in the kinematics model is not completely defined. The higher the valuation number, the greater the benefit from changing the deviations of the measuring points by adjusting the transformation. The valuation numbers are independent of the measured errors. They are defined by the kinematics model, the position and the number of measuring points per rotary axis.

The valuation of any rotary axis should not fall below a value of 2. Values greater or equal to 4 are desirable.

If the valuation numbers are too small, increase the measurement range of the rotary axis, or also the number of measuring points. If these measures do not improve the valuation number it may be because of an incorrect kinematic description. If necessary, inform your service agency.
Measurement uncertainty of angles

The TNC always indicates measurement uncertainty in degrees per 1 µm of system uncertainty. This information is important for evaluating the quality of the measured positioning errors, or the backlash of a rotary axis.

The system uncertainty includes at least the repeatability of the axes (backlash) as well as the positioning uncertainty of the linear axes (positioning errors) and of the touch probe. Since the TNC does not know the accuracy of the complete system, you must make a separate evaluation.

- Example of uncertainty of the calculated positioning errors:
  - Positioning uncertainty of each linear axis: 10 µm
  - Uncertainty of touch probe: 2 µm
  - Logged measurement uncertainty: 0.0002 °/µm
  - System uncertainty = \( \sqrt{3 \times 10^2 + 2^2} \) = 17.4 µm
  - Measurement uncertainty = 0.0002 °/µm * 17.4 µm = 0.0034°

- Example of uncertainty of the calculated backlash:
  - Repeatability of each linear axis: 5 µm
  - Uncertainty of touch probe: 2 µm
  - Logged measurement uncertainty: 0.0002 °/µm
  - System uncertainty = \( \sqrt{3 \times 5^2 + 2^2} \) = 8.9 µm
  - Measurement uncertainty = 0.0002 °/µm * 8.9 µm = 0.0018°
18.5 PRESET COMPENSATION
(Cycle 452, DIN/ISO: G452, Option)

Cycle run

Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see “MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451; Option)” on page 466). 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 tool changer 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 tool changer head up to the point where the head is changed
5. Use Cycle 452 to adjust other tool changer 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. Set 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.

<table>
<thead>
<tr>
<th>Parameter number</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q141</td>
<td>Standard deviation measured in the A axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q142</td>
<td>Standard deviation measured in the B axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q143</td>
<td>Standard deviation measured in the C axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q144</td>
<td>Optimized standard deviation in the A axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q145</td>
<td>Optimized standard deviation in the B axis (–1 if axis was not measured)</td>
</tr>
<tr>
<td>Q146</td>
<td>Optimized standard deviation in the C axis (–1 if axis was not measured)</td>
</tr>
</tbody>
</table>
Please note while programming:

- In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine tool manual provides further information.

- Note that all functions for tilting in the working plane are reset. M128 or 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 machine parameter MP6150, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate Q253, while the probe monitoring is inactive.

- If the kinematic data attained in the optimize mode are greater than the permissible limit (MP6000), 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 MP6601, the TNC shows an error message and ends the measurement.

- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

- Programming in inches: The TNC always records the log data and results of measurement in millimeters.
Cycle parameters

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999

- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999, alternatively **PREDEF**

- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the series A, then B, then C.
  - Input >0:
    Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. The probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool during positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**

- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000

- **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999

- **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999

- **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

- **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

- **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999

---

**Example: Calibration program**

```
4 TOOL CALL "TASTER" Z
5 TCH PROBE 450 SAVE KINEMATICS
Q410=0 ;MODE
Q409=5 ;MEMORY
6 TCH PROBE 452 PRESET COMPENSATION
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SETUP 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 MEAS. POINTS
Q432=0 ;BACKLASH, ANG. RANGE
```
- **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999

- **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

- **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis. If input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999

- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999

- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. If input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the calibration sphere in the plane with 4 or 3 probing points. 3 probing points increase the measuring speed:
  - **4**: Use 4 measuring points (standard setting)
  - **3**: Use 3 measuring points

- **Backlash, angle range** Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range: -3.0000 to +3.0000
## Adjustment of tool changer 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 tool changer 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 Q432 = 2 or 3 in Cycle 451) after measuring the reference head.

### Example: Measuring a reference head

```
1 TOOL CALL "TASTER" 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 MEAS. POINTS
Q431=3 ;PRESET
Q432=0 ;BACKLASH, ANG. RANGE
```
Insert the second tool changer head.
Insert the touch probe.
Measure the 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 tool changer 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.

Example: Adjusting a tool changer head

3 TOOL CALL "TASTER" 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 MEAS. POINTS
Q432=0 ;BACKLASH, ANG. RANGE
Drift Compensation

During machining various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle 452.

- Clamp the calibration sphere.
- Insert the touch probe.
- Measure the complete kinematics with Cycle 451 before starting the machining process.
- Set the preset (using Q432 = 2 or 3 in Cycle 451) after measuring the kinematics.
- Then set the presets on your workpieces and start the machining process.

Example: Reference measurement for drift compensation

```plaintext
1 TOOL CALL "TASTER" 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 ;INCL. ANGLE A AXIS
   Q414=4 ;MEAS. POINTS A AXIS
   Q415=-90 ;START ANGLE B AXIS
   Q416=+90 ;END ANGLE B AXIS
   Q417=0 ;INCL. ANGLE B AXIS
   Q418=2 ;MEAS. POINTS B AXIS
   Q419=+90 ;START ANGLE C AXIS
   Q420=+270 ;END ANGLE C AXIS
   Q421=0 ;INCL. ANGLE C AXIS
   Q422=3 ;MEAS. POINTS C AXIS
   Q423=4 ;NO. OF MEAS. POINTS
   Q431=3 ;PRESET
   Q432=0 ;BACKLASH, ANG. RANGE
```
Measure the drift of the axes at regular intervals.
- Insert the touch probe.
- Activate the preset in the calibration sphere.
- Use Cycle 452 to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the complete process.

This procedure can also be performed on machines without rotary axes.

Example: Drift compensation

```
4 TOOL CALL "TASTER" Z
5 TCH PROBE 452 PRESET COMPENSATION
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SET-UP CLEARANCE
Q408=0 ;RETR. HEIGHT
Q253=99999;F PRE-POSITIONING
Q380=45 ;REFERENCE ANGLE
Q411=-90 ;START ANGLE A AXIS
Q412=+90 ;END ANGLE A AXIS
Q413=45 ;INCID. ANGLE A AXIS
Q414=4 ;MEAS. POINTS A AXIS
Q415=-90 ;START ANGLE B AXIS
Q416=+90 ;END ANGLE B AXIS
Q417=0 ;INCID. ANGLE B AXIS
Q418=2 ;MEAS. POINTS B AXIS
Q419=+90 ;START ANGLE C AXIS
Q420=+270 ;END ANGLE C AXIS
Q421=0 ;INCID. ANGLE C AXIS
Q422=3 ;MEAS. POINTS C AXIS
Q423=3 ;NO. OF MEAS. POINTS
Q432=0 ;BACKLASH, ANG. RANGE
```
Log 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
  - Ascertained backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Evaluation of measuring points
  - Measurement uncertainty of rotary axes

Notes on log data

(see “Notes on log data” on page 480)
18.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, Option)
Touch Probe Cycles: Automatic Tool Measurement
19.1 Fundamentals

Overview

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 tool manual.

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.
- Measuring individual teeth.

You can program the cycles for tool measurement in the Programming and Editing mode of operation via the TOUCH PROBE key. The following cycles are available:

<table>
<thead>
<tr>
<th>Cycle</th>
<th>New format</th>
<th>Old format</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calibrating the TT, Cycles 30 and 480</td>
<td></td>
<td></td>
<td>Page 499</td>
</tr>
<tr>
<td>Calibrating the wireless TT 449, Cycle 484</td>
<td>449</td>
<td>484</td>
<td></td>
</tr>
<tr>
<td>Measuring the tool length, Cycles 31 and 481</td>
<td>31</td>
<td>31</td>
<td>Page 501</td>
</tr>
<tr>
<td>Measuring the tool radius, Cycles 32 and 482</td>
<td>32</td>
<td>32</td>
<td>Page 503</td>
</tr>
<tr>
<td>Measuring the tool length and radius, Cycles 33 and 483</td>
<td>33</td>
<td>33</td>
<td>Page 505</td>
</tr>
</tbody>
</table>

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.

You can also measure tools in a tilted working plane.
Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in controls for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter Q199.

Setting the machine parameters

The TNC uses the feed rate for probing defined in MP6520 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:

\[
    n = \frac{MP6570}{(r \times 0.0063)}
\]

- \( n \) Spindle speed [rpm]
- \( MP6570 \) Maximum permissible cutting speed in m/min
- \( r \) Active tool radius in mm

The feed rate for probing is calculated from:

\[
    v = \text{meas. tolerance} \times n
\]

- \( v \) Feed rate for probing in mm/min
- Measuring tolerance Measuring tolerance [mm], depending on MP6507
- \( n \) Speed in rpm
MP6507 determines the calculation of the probing feed rate:

**MP6507=0:**
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 (MP6570) and the permissible tolerance (MP6510), the sooner you will encounter this effect.

**MP6507=1:**
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:

<table>
<thead>
<tr>
<th>Tool radius</th>
<th>Measuring tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Up to 30 mm</td>
<td>MP6510</td>
</tr>
<tr>
<td>30 to 60 mm</td>
<td>2 * MP6510</td>
</tr>
<tr>
<td>60 to 90 mm</td>
<td>3 * MP6510</td>
</tr>
<tr>
<td>90 to 120 mm</td>
<td>4 * MP6510</td>
</tr>
</tbody>
</table>

**MP6507=2:**
The feed rate for probing remains constant, the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance = \( r \cdot MP6510/5 \) mm, where

- \( r \) Active tool radius in mm
- MP6510 Maximum permissible error of measurement
## Entries in the tool table TOOL.T

<table>
<thead>
<tr>
<th>Abbr.</th>
<th>Inputs</th>
<th>Dialog</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CUT</strong></td>
<td>Number of teeth (20 teeth maximum)</td>
<td><strong>Number of teeth?</strong></td>
</tr>
<tr>
<td><strong>LTOL</strong></td>
<td>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</td>
<td><strong>Wear tolerance: length?</strong></td>
</tr>
<tr>
<td><strong>RTOL</strong></td>
<td>Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status I). Input range: 0 to 0.9999 mm</td>
<td><strong>Wear tolerance: radius?</strong></td>
</tr>
<tr>
<td><strong>DIRECT.</strong></td>
<td>Cutting direction of the tool for measuring the tool during rotation</td>
<td><strong>Cutting direction (M3 = –)?</strong></td>
</tr>
<tr>
<td><strong>TT:R-OFFS</strong></td>
<td>Tool length measurement: Tool offset between stylus center and tool center. Preset value: Tool radius R (NO ENT means R).</td>
<td><strong>Tool offset: radius?</strong></td>
</tr>
<tr>
<td><strong>TT:L-OFFS</strong></td>
<td>Radius measurement: tool offset in addition to MP6530 between upper surface of stylus and lower surface of tool. Default: 0</td>
<td><strong>Tool offset: length?</strong></td>
</tr>
<tr>
<td><strong>LBREAK</strong></td>
<td>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</td>
<td><strong>Breakage tolerance: length?</strong></td>
</tr>
<tr>
<td><strong>RBREAK</strong></td>
<td>Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status I). Input range: 0 to 0.9999 mm</td>
<td><strong>Breakage tolerance: radius?</strong></td>
</tr>
</tbody>
</table>

### Input examples for common tool types

<table>
<thead>
<tr>
<th>Tool type</th>
<th>CUT</th>
<th>TT:R-OFFS</th>
<th>TT:L-OFFS</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Drill</strong></td>
<td>– (no function)</td>
<td>0 (no offset required because tool tip is to be measured)</td>
<td></td>
</tr>
<tr>
<td><strong>End mill</strong> with diameter of &lt; 19 mm</td>
<td>4 (4 teeth)</td>
<td>0 (no offset required because tool diameter is smaller than the contact plate diameter of the TT)</td>
<td>0 (no additional offset required during radius measurement. Offset from MP6530 is used.)</td>
</tr>
<tr>
<td><strong>End mill</strong> with diameter of &gt; 19 mm</td>
<td>4 (4 teeth)</td>
<td>R (offset required because tool diameter is larger than the contact plate diameter of the TT)</td>
<td>0 (no additional offset required during radius measurement. Offset from MP6530 is used.)</td>
</tr>
<tr>
<td><strong>Radius cutter</strong></td>
<td>4 (4 teeth)</td>
<td>0 (no offset required because the south pole of the ball is to be measured)</td>
<td>5 (always define the tool radius as the offset so that the diameter is not measured in the radius)</td>
</tr>
</tbody>
</table>
Display of the measurement results

You can display the results of tool measurement in the additional status display (in the machine operating modes). The TNC then shows the program blocks in the left and the measuring results in the right screen window. The measuring results that exceed the permissible wear tolerance are marked in the status display with an asterisk “*”; the results that exceed the permissible breakage tolerance are marked with the character B.
19.2 Calibrating the TT (Cycle 30 or 480, DIN/ISO: G480)

Cycle run

The TT is calibrated with the measuring cycle TCH PROBE 30 or TCH PROBE 480 (see also “Differences between Cycles 31 to 33 and Cycles 481 to 483” on page 495). 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 measurement.

The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. This constellation causes a deformation of 0.1 µm per 1 N probing force.

Please note while programming:

The functioning of the calibration cycle is dependent on MP 6500. 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 6580.0 to 6580.2.

If you change the setting of any of the Machine Parameters 6580.0 to 6580.2, you must recalibrate the TT.

Cycle parameters

➤ Clearance height: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the calibration tool above the level of the probe contact (safety zone from MP6540). Input range -99999.9999 to 99999.9999, alternatively PREDEF

Example: NC blocks in old format

```
6 TOOL CALL 1 Z
7 TCH PROBE 30.0 CALIBRATE TT
8 TCH PROBE 30.1 HEIGHT: +90
```

Example: NC blocks in new format

```
6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
Q260=+100 ;CLEARANCE HEIGHT
```
19.3 CALIBRATING THE WIRELESS TT 449 (Cycle 484, DIN/ISO: G484)

Fundamentals
With Cycle 484, you calibrate the wireless infrared TT 449 tool touch probe. The calibration process is not completely automated, because the TT’s position on the table is not defined.

Cycle run
- Insert the calibrating tool.
- Define and start the calibration cycle.
- Position the calibrating tool manually above the center of the touch probe and follow the instructions in the pop-up window. Ensure that the calibrating tool is located above the measuring surface of the probe contact.

The calibration process is semi-automatic. 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 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 measurement.

Please note while programming:
- The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. This constellation causes a deformation of 0.1 µm per 1 N probing force.

Cycle parameters
Cycle 484 has no cycle parameters.
19.4 Measuring the 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 also “Differences between Cycles 31 to 33 and Cycles 481 to 483” on page 495). 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 can 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 can 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 can 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 system and then moving it toward the measuring surface until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (TT: R-OFFS).

Cycle for measuring a tool during standstill (e.g. for drills)

The control positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until it touches the surface. To activate this function, enter zero for the Tool offset: Radius (TT: R-OFFS) in the tool table.

Cycle for measuring individual teeth

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in MP6530. You can enter an additional offset with Tool offset: Length (TT: L-OFFS) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for CUTTER MEASUREMENT.
Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

You can run an individual tooth measurement of tools with up to 99 teeth. The TNC shows the measured values of up to 24 teeth in the status display.

Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

- **Parameter number for result?**: Parameter number in which the TNC stores the status of the measurement:
  - 0.0: Tool is within the 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 MP6540). Input range -99999.9999 to 99999.9999, alternatively PREDEF

- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is to measure the individual teeth (maximum of 99 teeth)

**Example: Measuring a rotating tool for the first time; old format**

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0
```

**Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format**

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 Q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1
```

**Example: NC blocks in new format**

```
6 TOOL CALL 12 Z
7 TCH PROBE 481 TOOL LENGTH
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH
```
19.5 Measuring the Tool Radius
(Cycle 32 or 482, ISO: G482)

Cycle run

To measure the tool radius, program the cycle TCH PROBE 32 or TCH PROBE 482 (see also “Differences between Cycles 31 to 33 and Cycles 481 to 483” on page 495). Via input parameters you can measure the radius of a tool by two methods:

- 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 MP6530. 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 the number of teeth (CUT) with 0 and adjust the machine parameter 6500. Refer to your machine tool manual.

> You can run an individual tooth measurement of tools with up to 99 teeth. The TNC shows the measured values of up to 24 teeth in the status display.
19.5 Measuring the Tool Radius (Cycle 32 or 482, ISO: G482)

**Cycle parameters**

- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  - 0.0: Tool is within the 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 MP6540). Input range -99999.9999 to 99999.9999, alternatively PREDEF

- **Cutter measurement? 0=No / 1=Yes:** Choose whether the control is also to measure the individual teeth (maximum of 99 teeth)

---

**Example: Measuring a rotating tool for the first time; old format**

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0
```

**Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format**

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 Q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1
```

**Example: NC blocks in new format**

```
6 TOOL CALL 12 Z
7 TCH PROBE 482 TOOL RADIUS
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH
```
19.6 Measuring Tool Length and Radius (Cycle 33 or 483, ISO: G483)

Cycle run

To measure both the length and radius of a tool, program the measuring cycle TCH PROBE 33 or TCH PROBE 482 (see also “Differences between Cycles 31 to 33 and Cycles 481 to 483” on page 495). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. In 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 measuring cycles 31 and 32.

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 the number of teeth (CUT) with 0 and adjust the machine parameter 6500. Refer to your machine tool manual.

You can run an individual tooth measurement of tools with up to 99 teeth. The TNC shows the measured values of up to 24 teeth in the status display.
Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius $R$ and the tool length $L$ in the central tool file TOOL.T by the delta values $DR = 0$ and $DL = 0$. If you wish to inspect a tool, the TNC compares the measured data with the tool data stored in TOOL.T. The TNC calculates the deviations and enters them as positive or negative delta values $DR$ and $DL$ in TOOL.T. The deviations are also available in the Q parameters Q115 and Q116. If the delta values are greater than the permissible tool tolerances for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

- **Parameter number for result?**: Parameter number in which the TNC stores the status of the measurement:
  - **0.0**: Tool is within the tolerance
  - **1.0**: Tool is worn ($LTOL$ or/and $RTOL$ exceeded)
  - **2.0**: Tool is broken ($LBREAK$ or/and $RBREAK$ 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 MP6540).

  - **Input range -99999.9999 to 99999.9999**, alternatively **PREDEF**

- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is also to measure the individual teeth (maximum of 99 teeth)

### Example: 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
```

### Example: 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
```

### Example: 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
```
Symbole
3-D touch probes ... 44, 312
  Calibrating
    Triggering ... 447, 448

A
Angle of a plane, measuring ... 437
Angle, measuring in a plane ... 437
Automatic tool measurement ... 497
Axis-specific scaling ... 288

B
Back boring ... 87
Basic rotation
  Measuring during program run ... 320
  Setting directly ... 335
Bolt hole circle ... 173
Bolt hole circle, measuring ... 433
Bore milling ... 95
Boring ... 79

C
Calibrate touch probe
  automatically ... 458
Centering ... 73
Circle, measuring from inside ... 408
Circle, measuring from outside ... 412
Circular pocket
  Roughing+finishing ... 146
Circular slot
  Roughing+finishing ... 155
Circular stud ... 164
Classification of results ... 399
Compensating workpiece misalignment
  By measuring two points of a line ... 322
  Over two holes ... 325
  Over two studs ... 328
  Via rotary axis ... 331, 336
Confidence range ... 316
Contour cycles ... 182
Contour train ... 201
Contour train data ... 203
Coordinate transformation ... 272
Coordinate, measuring a single ... 430
Cycle
  Calling ... 50
  Defining ... 49

C
Cycles and point tables ... 69
Cylinder surface
  Contour machining ... 221
  Contour milling ... 230
  Ridge machining ... 227
  Slot machining ... 224

D
Datum
  Save in a datum table ... 344
  Save in the preset table ... 344
Datum shift
  With datum tables ... 275
  Within the program ... 274
Deepened starting point for drilling ... 94, 99
Drilling ... 75, 83, 91
  Deepened starting point ... 94, 99
Drilling cycles ... 72
Dwell time ... 303

E
External thread milling ... 132

F
Face milling ... 263
Fast probing ... 456
FCL function ... 8
Feature content level ... 8
Floor finishing ... 198

G
Global settings ... 456

H
Hard milling ... 205
Helical thread drilling/milling ... 128
Hole, measuring ... 408

K
Kinematic measurement ... 462, 466
  Accuracy ... 471
  Backlash ... 473
  Calibration methods ... 472, 487, 489
  Hirth coupling ... 469
  Kinematic measurement ... 466, 482
  Kinematics, save ... 464
  Log function ... 465, 479, 491
  Measuring points, choice of ... 470
  Measuring positions, choice of ... 470
  Prerequisites ... 463
  Preset compensation ... 482
  KinematicsOpt ... 462

M
Machine parameters for 3-D touch probes ... 315
Machining patterns ... 58
Measurement results in Q parameters ... 344, 399
Measuring angles ... 405
Mirror image ... 282
Multiple measurements ... 316

O
Oriented spindle stop ... 306

P
Pattern definition ... 58
Pecking ... 91, 98
  Deepened starting point ... 94, 99
Point pattern
  Circular ... 173
  Linear ... 176
  Overview ... 172
Point patterns
Point tables ... 66
Positioning logic ... 318
Preset table ... 344
Index

P
Presetting automatically ... 342
Center of 4 holes ... 383
Center of bolt hole circle ... 377
Center of circular pocket (or hole) ... 360
Center of circular stud ... 364
Center of rectangular pocket ... 352
Center of rectangular stud ... 356
In any axis ... 387
In inside corner ... 373
In the touch probe axis ... 381
Outside corner ... 368
Ridge center ... 349
Slot center ... 345
Probing feed rate ... 317
Program call
Via cycle ... 304

R
Reaming ... 77
Recording the results of measurement ... 397
Rectangular pocket
  Roughing+finishing ... 141
Rectangular pocket measurement ... 420
Rectangular stud ... 160
Rectangular stud, measuring ... 416
Result parameters ... 344, 399
Ridge, measuring from outside ... 427
Rotation ... 284
Rough out: See SL Cycles: Rough-out
Ruled surface ... 259
Run 3-D data ... 255

S
Scaling factor ... 286
Side finishing ... 199
Single-fluted deep-hole drilling ... 98
SL Cycles
SL cycles
  Basics ... 182, 249
  Contour data ... 190
  Contour geometry cycle ... 185
  Contour train ... 201
  Contour train data ... 203
  Floor finishing ... 198
  Overlapping contours ... 186, 243
  Pilot drilling ... 192
  Rough-out ... 194
  Side finishing ... 199
SL cycles with complex contour formula ... 238
SL cycles with simple contour formula ... 249
Slot milling
  Contour slot ... 205
  Roughing-finishing ... 150
Slot width, measuring ... 424

T
Tapping
  With a floating tap holder ... 107
  With chip breaking ... 112
  Without floating tap holder ... 109, 112
Thermal expansion, measuring ... 453
Thread drilling/milling ... 124
Thread milling, fundamentals ... 115
Thread milling, internal ... 117
Thread milling/countersinking ... 120
Tilting the working plane ... 290
  Cycle ... 290
  Guide ... 297
Tolerance monitoring ... 400
Tool compensation ... 400
Tool measurement ... 497
  Calibrating the TT ... 499, 500
  Displaying the measurement results ... 498
  Machine parameters ... 495
  Measuring tool length and radius ... 505
  Tool length ... 501
  Tool radius ... 503
Tool monitoring ... 400
Touch probe cycles
  Touch probe cycles for automatic operation ... 314
Trochoidal milling ... 205

U
Universal drilling ... 83, 91

W
Width, measuring from inside ... 424
Width, measuring from outside ... 427
Workpiece measurement ... 396
# Overview

## Fixed cycles

<table>
<thead>
<tr>
<th>Cycle number</th>
<th>Cycle designation</th>
<th>DEF active</th>
<th>CALL active</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>Datum shift</td>
<td></td>
<td></td>
<td>Page 274</td>
</tr>
<tr>
<td>8</td>
<td>Mirror image</td>
<td></td>
<td></td>
<td>Page 282</td>
</tr>
<tr>
<td>9</td>
<td>Dwell time</td>
<td></td>
<td></td>
<td>Page 303</td>
</tr>
<tr>
<td>10</td>
<td>Rotation</td>
<td></td>
<td></td>
<td>Page 284</td>
</tr>
<tr>
<td>11</td>
<td>Scaling factor</td>
<td></td>
<td></td>
<td>Page 286</td>
</tr>
<tr>
<td>12</td>
<td>Program call</td>
<td></td>
<td></td>
<td>Page 304</td>
</tr>
<tr>
<td>13</td>
<td>Oriented spindle stop</td>
<td></td>
<td></td>
<td>Page 306</td>
</tr>
<tr>
<td>14</td>
<td>Contour definition</td>
<td></td>
<td></td>
<td>Page 185</td>
</tr>
<tr>
<td>19</td>
<td>Tilting the working plane</td>
<td></td>
<td></td>
<td>Page 290</td>
</tr>
<tr>
<td>20</td>
<td>SL II contour data</td>
<td></td>
<td></td>
<td>Page 190</td>
</tr>
<tr>
<td>21</td>
<td>SL II pilot drilling</td>
<td></td>
<td></td>
<td>Page 192</td>
</tr>
<tr>
<td>22</td>
<td>SL II rough out</td>
<td></td>
<td></td>
<td>Page 194</td>
</tr>
<tr>
<td>23</td>
<td>Floor finishing SL II</td>
<td></td>
<td></td>
<td>Page 198</td>
</tr>
<tr>
<td>24</td>
<td>Side finishing SL II</td>
<td></td>
<td></td>
<td>Page 199</td>
</tr>
<tr>
<td>25</td>
<td>Contour train</td>
<td></td>
<td></td>
<td>Page 201</td>
</tr>
<tr>
<td>26</td>
<td>Axis-specific scaling</td>
<td></td>
<td></td>
<td>Page 288</td>
</tr>
<tr>
<td>27</td>
<td>Cylinder surface</td>
<td></td>
<td></td>
<td>Page 221</td>
</tr>
<tr>
<td>28</td>
<td>Cylindrical surface slot</td>
<td></td>
<td></td>
<td>Page 224</td>
</tr>
<tr>
<td>29</td>
<td>Cylinder surface ridge</td>
<td></td>
<td></td>
<td>Page 227</td>
</tr>
<tr>
<td>30</td>
<td>Run 3-D data</td>
<td></td>
<td></td>
<td>Page 255</td>
</tr>
<tr>
<td>32</td>
<td>Tolerance</td>
<td></td>
<td></td>
<td>Page 307</td>
</tr>
<tr>
<td>39</td>
<td>Cylinder surface external contour</td>
<td></td>
<td></td>
<td>Page 230</td>
</tr>
<tr>
<td>200</td>
<td>Drilling</td>
<td></td>
<td></td>
<td>Page 75</td>
</tr>
<tr>
<td>201</td>
<td>Reaming</td>
<td></td>
<td></td>
<td>Page 77</td>
</tr>
<tr>
<td>202</td>
<td>Boring</td>
<td></td>
<td></td>
<td>Page 79</td>
</tr>
<tr>
<td>203</td>
<td>Universal drilling</td>
<td></td>
<td></td>
<td>Page 83</td>
</tr>
<tr>
<td>Cycle number</td>
<td>Cycle designation</td>
<td>DEF active</td>
<td>CALL active</td>
<td>Page</td>
</tr>
<tr>
<td>--------------</td>
<td>------------------------------------------------</td>
<td>------------</td>
<td>-------------</td>
<td>---------</td>
</tr>
<tr>
<td>204</td>
<td>Back boring</td>
<td></td>
<td></td>
<td>Page 87</td>
</tr>
<tr>
<td>205</td>
<td>Universal pecking</td>
<td></td>
<td></td>
<td>Page 91</td>
</tr>
<tr>
<td>206</td>
<td>Tapping with a floating tap holder, new</td>
<td></td>
<td></td>
<td>Page 107</td>
</tr>
<tr>
<td>207</td>
<td>Rigid tapping, new</td>
<td></td>
<td></td>
<td>Page 109</td>
</tr>
<tr>
<td>208</td>
<td>Bore milling</td>
<td></td>
<td></td>
<td>Page 95</td>
</tr>
<tr>
<td>209</td>
<td>Tapping with chip breaking</td>
<td></td>
<td></td>
<td>Page 112</td>
</tr>
<tr>
<td>220</td>
<td>Circular point pattern</td>
<td></td>
<td></td>
<td>Page 173</td>
</tr>
<tr>
<td>221</td>
<td>Linear point pattern</td>
<td></td>
<td></td>
<td>Page 176</td>
</tr>
<tr>
<td>230</td>
<td>Multipass milling</td>
<td></td>
<td></td>
<td>Page 257</td>
</tr>
<tr>
<td>231</td>
<td>Ruled surface</td>
<td></td>
<td></td>
<td>Page 259</td>
</tr>
<tr>
<td>232</td>
<td>Face milling</td>
<td></td>
<td></td>
<td>Page 263</td>
</tr>
<tr>
<td>240</td>
<td>Centering</td>
<td></td>
<td></td>
<td>Page 73</td>
</tr>
<tr>
<td>241</td>
<td>Single-fluted deep-hole drilling</td>
<td></td>
<td></td>
<td>Page 98</td>
</tr>
<tr>
<td>247</td>
<td>Datum setting</td>
<td></td>
<td></td>
<td>Page 281</td>
</tr>
<tr>
<td>251</td>
<td>Rectangular pocket (complete machining)</td>
<td></td>
<td></td>
<td>Page 141</td>
</tr>
<tr>
<td>252</td>
<td>Circular pocket (complete machining)</td>
<td></td>
<td></td>
<td>Page 146</td>
</tr>
<tr>
<td>253</td>
<td>Slot milling</td>
<td></td>
<td></td>
<td>Page 150</td>
</tr>
<tr>
<td>254</td>
<td>Circular slot</td>
<td></td>
<td></td>
<td>Page 155</td>
</tr>
<tr>
<td>256</td>
<td>Rectangular stud (complete machining)</td>
<td></td>
<td></td>
<td>Page 160</td>
</tr>
<tr>
<td>257</td>
<td>Circular stud (complete machining)</td>
<td></td>
<td></td>
<td>Page 164</td>
</tr>
<tr>
<td>262</td>
<td>Thread milling</td>
<td></td>
<td></td>
<td>Page 117</td>
</tr>
<tr>
<td>263</td>
<td>Thread milling/countersinking</td>
<td></td>
<td></td>
<td>Page 120</td>
</tr>
<tr>
<td>264</td>
<td>Thread drilling/milling</td>
<td></td>
<td></td>
<td>Page 124</td>
</tr>
<tr>
<td>265</td>
<td>Helical thread drilling/milling</td>
<td></td>
<td></td>
<td>Page 128</td>
</tr>
<tr>
<td>267</td>
<td>Outside thread milling</td>
<td></td>
<td></td>
<td>Page 132</td>
</tr>
<tr>
<td>270</td>
<td>Contour train data</td>
<td></td>
<td></td>
<td>Page 203</td>
</tr>
<tr>
<td>275</td>
<td>Trochoidal slot</td>
<td></td>
<td></td>
<td>Page 205</td>
</tr>
</tbody>
</table>
## Touch probe cycles

<table>
<thead>
<tr>
<th>Cycle number</th>
<th>Cycle designation</th>
<th>DEF active</th>
<th>CALL active</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Reference plane</td>
<td></td>
<td></td>
<td>Page 402</td>
</tr>
<tr>
<td>1</td>
<td>Polar datum</td>
<td></td>
<td></td>
<td>Page 403</td>
</tr>
<tr>
<td>2</td>
<td>Calibrate TS radius</td>
<td></td>
<td></td>
<td>Page 447</td>
</tr>
<tr>
<td>3</td>
<td>Measuring</td>
<td></td>
<td></td>
<td>Page 449</td>
</tr>
<tr>
<td>4</td>
<td>Measuring in 3-D</td>
<td></td>
<td></td>
<td>Page 451</td>
</tr>
<tr>
<td>9</td>
<td>Calibrate TS length</td>
<td></td>
<td></td>
<td>Page 448</td>
</tr>
<tr>
<td>30</td>
<td>Calibrating the TT</td>
<td></td>
<td></td>
<td>Page 499</td>
</tr>
<tr>
<td>31</td>
<td>Measure/Inspect the tool length</td>
<td></td>
<td></td>
<td>Page 501</td>
</tr>
<tr>
<td>32</td>
<td>Measure/Inspect the tool radius</td>
<td></td>
<td></td>
<td>Page 503</td>
</tr>
<tr>
<td>33</td>
<td>Measure/Inspect the tool length and the tool radius</td>
<td></td>
<td></td>
<td>Page 505</td>
</tr>
<tr>
<td>400</td>
<td>Basic rotation using two points</td>
<td></td>
<td></td>
<td>Page 322</td>
</tr>
<tr>
<td>401</td>
<td>Basic rotation from two holes</td>
<td></td>
<td></td>
<td>Page 325</td>
</tr>
<tr>
<td>402</td>
<td>Basic rotation from two studs</td>
<td></td>
<td></td>
<td>Page 328</td>
</tr>
<tr>
<td>403</td>
<td>Compensate misalignment with rotary axis</td>
<td></td>
<td></td>
<td>Page 331</td>
</tr>
<tr>
<td>404</td>
<td>Set basic rotation</td>
<td></td>
<td></td>
<td>Page 335</td>
</tr>
<tr>
<td>405</td>
<td>Compensate misalignment with the C axis</td>
<td></td>
<td></td>
<td>Page 336</td>
</tr>
<tr>
<td>408</td>
<td>Reference point at slot center (FCL 3 function)</td>
<td></td>
<td></td>
<td>Page 345</td>
</tr>
<tr>
<td>409</td>
<td>Reference point at ridge center (FCL 3 function)</td>
<td></td>
<td></td>
<td>Page 349</td>
</tr>
<tr>
<td>410</td>
<td>Datum from inside of rectangle</td>
<td></td>
<td></td>
<td>Page 352</td>
</tr>
<tr>
<td>411</td>
<td>Datum from outside of rectangle</td>
<td></td>
<td></td>
<td>Page 356</td>
</tr>
<tr>
<td>412</td>
<td>Datum from inside of circle (hole)</td>
<td></td>
<td></td>
<td>Page 360</td>
</tr>
<tr>
<td>413</td>
<td>Datum from outside of circle (stud)</td>
<td></td>
<td></td>
<td>Page 364</td>
</tr>
<tr>
<td>414</td>
<td>Datum from outside of corner</td>
<td></td>
<td></td>
<td>Page 368</td>
</tr>
<tr>
<td>415</td>
<td>Datum from inside of corner</td>
<td></td>
<td></td>
<td>Page 373</td>
</tr>
<tr>
<td>416</td>
<td>Datum from circle center</td>
<td></td>
<td></td>
<td>Page 377</td>
</tr>
<tr>
<td>417</td>
<td>Datum in touch probe axis</td>
<td></td>
<td></td>
<td>Page 381</td>
</tr>
<tr>
<td>418</td>
<td>Datum at center between four holes</td>
<td></td>
<td></td>
<td>Page 383</td>
</tr>
<tr>
<td>419</td>
<td>Datum in any one axis</td>
<td></td>
<td></td>
<td>Page 387</td>
</tr>
<tr>
<td>Cycle number</td>
<td>Cycle designation</td>
<td>DEF active</td>
<td>CALL active</td>
<td>Page</td>
</tr>
<tr>
<td>-------------</td>
<td>------------------------------------------------------------</td>
<td>------------</td>
<td>-------------</td>
<td>-------</td>
</tr>
<tr>
<td>420</td>
<td>Workpiece—measure angle</td>
<td></td>
<td></td>
<td>Page 405</td>
</tr>
<tr>
<td>421</td>
<td>Workpiece—measure hole (center and diameter of hole)</td>
<td></td>
<td></td>
<td>Page 408</td>
</tr>
<tr>
<td>422</td>
<td>Workpiece—measure circle from outside (diameter of circular stud)</td>
<td></td>
<td></td>
<td>Page 412</td>
</tr>
<tr>
<td>423</td>
<td>Workpiece—measure rectangle from inside</td>
<td></td>
<td></td>
<td>Page 412</td>
</tr>
<tr>
<td>424</td>
<td>Workpiece—measure rectangle from outside</td>
<td></td>
<td></td>
<td>Page 420</td>
</tr>
<tr>
<td>425</td>
<td>Workpiece—measure inside width (slot)</td>
<td></td>
<td></td>
<td>Page 424</td>
</tr>
<tr>
<td>426</td>
<td>Workpiece—measure outside width (ridge)</td>
<td></td>
<td></td>
<td>Page 427</td>
</tr>
<tr>
<td>427</td>
<td>Workpiece—measure in any selectable axis</td>
<td></td>
<td></td>
<td>Page 430</td>
</tr>
<tr>
<td>430</td>
<td>Workpiece—measure bolt hole circle</td>
<td></td>
<td></td>
<td>Page 433</td>
</tr>
<tr>
<td>431</td>
<td>Workpiece—measure plane</td>
<td></td>
<td></td>
<td>Page 433</td>
</tr>
<tr>
<td>440</td>
<td>Measure axis shift</td>
<td></td>
<td></td>
<td>Page 453</td>
</tr>
<tr>
<td>441</td>
<td>Rapid probing: Set global touch probe parameters (FCL 2 function)</td>
<td></td>
<td></td>
<td>Page 456</td>
</tr>
<tr>
<td>450</td>
<td>KinematicsOpt: Save kinematics (option)</td>
<td></td>
<td></td>
<td>Page 464</td>
</tr>
<tr>
<td>451</td>
<td>KinematicsOpt: Measure kinematics (option)</td>
<td></td>
<td></td>
<td>Page 466</td>
</tr>
<tr>
<td>452</td>
<td>KinematicsOpt: Preset compensation (option)</td>
<td></td>
<td></td>
<td>Page 466</td>
</tr>
<tr>
<td>460</td>
<td>Calibrate TS: Radius and length calibration on a calibration sphere</td>
<td></td>
<td></td>
<td>Page 458</td>
</tr>
<tr>
<td>480</td>
<td>Calibrate the TT</td>
<td></td>
<td></td>
<td>Page 499</td>
</tr>
<tr>
<td>481</td>
<td>Measure/Inspect the tool length</td>
<td></td>
<td></td>
<td>Page 501</td>
</tr>
<tr>
<td>482</td>
<td>Measure/Inspect the tool radius</td>
<td></td>
<td></td>
<td>Page 503</td>
</tr>
<tr>
<td>483</td>
<td>Measure/Inspect the tool length and the tool radius</td>
<td></td>
<td></td>
<td>Page 505</td>
</tr>
<tr>
<td>484</td>
<td>Calibrate infrared TT</td>
<td></td>
<td></td>
<td>Page 500</td>
</tr>
</tbody>
</table>
3-D Touch Probe Systems from HEIDENHAIN
help you to reduce non-cutting time:

For example in

- workpiece alignment
- datum setting
- workpiece measurement
- digitizing 3-D surfaces

with the workpiece touch probes

**TS 220** with cable
**TS 640** with infrared transmission

- tool measurement
- wear monitoring
- tool breakage monitoring

with the tool touch probe

**TT 140**