TNC 640
User’s manual for cycle programming

NC Software
340590-06
340591-06
340595-06

English (en)
9/2015
Fundamentals
About this Manual
The symbols used in this manual are described below.

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

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

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

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

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

Would you like any changes, or have you found any errors?
We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **tnc-userdoc@heidenhain.de**.
TNC model, software and features

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

<table>
<thead>
<tr>
<th>TNC model</th>
<th>NC software number</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC 640</td>
<td>340590-06</td>
</tr>
<tr>
<td>TNC 640 E</td>
<td>340591-06</td>
</tr>
<tr>
<td>TNC 640 Programming Station</td>
<td>340595-06</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

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

TNC functions that may not be available on your machine include:
- Tool measurement with the TT

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

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

User's Manual:

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


Software options

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

**Additional Axis (options 0 to 7)**

<table>
<thead>
<tr>
<th>Additional axis</th>
<th>Additional control loops 1 to 8</th>
</tr>
</thead>
</table>

**Advanced Function Set 1 (option 8)**

**Expanded functions Group 1**

- **Machining with rotary tables**
  - Cylindrical contours as if in two axes
  - Feed rate in distance per minute

**Coordinate transformations:**

- Tilting the working plane

**Interpolation:**

- Circle in 3 axes with tilted working plane (spatial arc)

**Advanced Function Set 2 (option 9)**

**Expanded functions Group 2**

- **3-D machining:**
  - Motion control with minimum jerk
  - 3-D tool compensation through surface normal vectors
  - Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point. (TCPM = Tool Center Point Management)
  - Keeping the tool normal to the contour
  - Tool radius compensation perpendicular to traversing direction and tool direction

**Interpolation:**

- Linear in 5 axes (subject to export permit)

**HEIDENHAIN DNC (option 18)**

Communication with external PC applications over COM component

**Display Step (option 23)**

**Display step**

**Input resolution:**

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

**Dynamic Collision Monitoring – DCM (option 40)**

**Dynamic Collision Monitoring**

- The machine manufacturer defines objects to be monitored
- Warning in Manual operation
- Program interrupt in Automatic operation
- Includes monitoring of 5-axis movements
### TNC model, software and features

**DXF Converter (option 42)**

DXF Converter
- Supported DXF format: AC1009 (AutoCAD R12)
- Adoption of contours and point patterns
- Simple and convenient specification of reference points
- Select graphical features of contour sections from conversational programs

**Adaptive Feed Control – AFC (option 45)**

Adaptive Feed Control
- Recording the actual spindle power by means of a teach-in cut
- Defining the limits of automatic feed rate control
- Fully automatic feed control during program run

**KinematicsOpt (option 48)**

Optimizing the machine kinematics
- Backup/restore active kinematics
- Test active kinematics
- Optimize active kinematics

**Mill-Turning (option 50)**

Milling and turning modes
- Functions:
  - Switching between Milling/Turning mode of operation
  - Constant surface speed
  - Tool-tip radius compensation
  - Turning cycles

**Extended Tool Management (option 93)**

Extended tool management
- Python-based

**Advanced Spindle Interpolation (option number 96)**

Interpolating spindle
- Interpolation turning:
  - Cycle 880: Gear hobbing
  - Cycle 291: Interpolation turning, coupling
  - Cycle 292: Interpolation turning, contour finishing

**Spindle Synchronism (option 131)**

Spindle synchronization
- Synchronization of milling spindle and turning spindle

**Remote Desktop Manager (option 133)**

Remote operation of external computer units
- Windows on a separate computer unit
- Incorporated in the TNC interface

**Synchronizing Functions (option 135)**

Synchronization functions
- Real Time Coupling – RTC:
  - Coupling of axes
### Visual Setup Control – VSC (option number 136)

<table>
<thead>
<tr>
<th>Camera-based monitoring of the setup situation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Record the setup situation with a HEIDENHAIN camera system</td>
</tr>
<tr>
<td>Visual comparison of planned and actual status in the workspace</td>
</tr>
</tbody>
</table>

### Cross Talk Compensation – CTC (option number 141)

<table>
<thead>
<tr>
<th>Compensation of axis couplings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Determination of dynamically caused position deviation through axis acceleration</td>
</tr>
<tr>
<td>Compensation of the TCP (Tool Center Point)</td>
</tr>
</tbody>
</table>

### Position Adaptive Control – PAC (option 142)

<table>
<thead>
<tr>
<th>Adaptive position control</th>
</tr>
</thead>
<tbody>
<tr>
<td>Changing of the control parameters depending on the position of the axes in the working space</td>
</tr>
<tr>
<td>Changing of the control parameters depending on the speed or acceleration of an axis</td>
</tr>
</tbody>
</table>

### Load Adaptive Control – LAC (option 143)

<table>
<thead>
<tr>
<th>Adaptive load control</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automatic determination of workpiece weight and frictional forces</td>
</tr>
<tr>
<td>Changing of control parameters depending on the actual mass of the workpiece</td>
</tr>
</tbody>
</table>

### Active Chatter Control – ACC (option number 145)

<table>
<thead>
<tr>
<th>Active chatter control</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fully automatic function for chatter control during machining</td>
</tr>
</tbody>
</table>

### Active Vibration Damping – AVD (option number 146)

<table>
<thead>
<tr>
<th>Active vibration damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>Damping of machine oscillations to improve the workpiece surface</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 upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.

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

Upgrade functions are identified in the manual with FCL n, where n indicates the sequential number of the feature content level. You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

**Intended place of operation**

The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

**Legal information**

This product uses open source software. Further information is available on the control under

- Programming and Editing operating mode
- MOD function
- LICENSE INFO softkey
Optional parameters

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

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

Compatibility

The majority of part programs created on older HEIDENHAIN contouring controls (TNC 150 B and higher) can be executed with this new software version of the TNC 640. Even if new, optional parameters ("Optional parameters") have been added to existing cycles, you can normally continue running your programs as usual. This is achieved by using the stored default value. The other way round, if a program created with a new software version is to be run on an older control, you can delete the respective optional Q parameters from the cycle definition with the NO ENT key. In this way you can ensure that the program will be downward compatible. If NC blocks contain invalid elements, the TNC will mark them as ERROR blocks when the file is opened.
New cycle functions of software 34059x-04

- The character set of the fixed cycle 225 Engraving was expanded by more characters and the diameter sign see "ENGRAVING (Cycle 225, DIN/ISO: G225)", page 309
- New machining cycle 275 Trochoidal milling see "TROCHOIDAL SLOT (Cycle 275, DIN ISO G275)", page 219
- New machining cycle 233 Face milling see "FACE MILLING (Cycle 233, DIN/ISO: G233)", page 174
- In Cycle 205 Universal Pecking you can now use parameter Q208 to define a feed rate for retraction see "Cycle parameters", page 94
- In the thread milling cycles 26x an approaching feed rate was introduced see "Cycle parameters", page 121
- The parameter Q305 NUMBER IN TABLE was added to Cycle 404 see "Cycle parameters", page 472
- In the drilling cycles 200, 203 and 205 the parameter Q395 DEPTH REFERENCE was introduced in order to evaluate the T ANGLE see "Cycle parameters", page 94
- Cycle 241 SINGLE-LIP DEEP HOLE DRILLING was expanded by several input parameters see "SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)", page 99
- The probing cycle 4 MEASURING IN 3-D was introduced see "MEASURING IN 3-D (Cycle 4)", page 583
New and changed cycle functions of software 34059x-05

- New Cycle 880 GEAR HOBBING (software option 50), see "GEAR HOBBING (Cycle 880, DIN/ISO: G880)", page 435
- New Cycle 292 CONTOUR FINISHING TURNING INTERPOLATION (software option 96), see "CONTOUR TURNING INTERPOLATION (Cycle 292, DIN/ISO: G292, software option 96)", page 294
- New Cycle 291 COUPLING TURNING INTERPOLATION (software option 96), see "COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)", page 303
- New Load Adaptive Control (LAC) cycle for the load-dependent adaptation of control parameters (software option 143), see "ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143)", page 318
- Cycle 270: CONTOUR TRAIN DATA was added to the cycle package (software option 19), see "CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)", page 218
- Cycle 39 CYLINDER SURFACE (software option 1) Contour was added to the cycle package, see "CYLINDER SURFACE (Cycle 39, DIN/ISO: G139, software option 1)", page 240
- The character set of the fixed cycle 225 Engraving was expanded by the CE, © and @ characters and the system time, see "ENGRAVING (Cycle 225, DIN/ISO: G225)", page 309
- Cycles 252 to 254 were expanded by the optional parameter Q439, see "Cycle parameters", page 150
- Cycle 22 was expanded by the optional parameters Q401 and Q404, see "ROUGHING (Cycle 22, DIN/ISO: G122)", page 207
- Cycles 841, 842, 851 and 852 were expanded by the plunging feed rate Q488, see "Cycle parameters", page 382
- Cycle 484 was expanded by the optional parameter Q536, see "Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484)", page 657
- Eccentric turning with Cycle 800 is possible with option 50, see "ADAPT ROTARY COORDINATE SYSTEM(Cycle 800, DIN/ISO: G800)", page 332
New and changed cycle functions of software 34059x-06

- New cycle 258 POLYGON STUD see "CIRCULAR STUD (cycle 258, DIN/ISO: G258)", page 169
- New cycles 600 and 601 for Visual Setup Control (software option 136), see "Camera-based monitoring of the setup situation VSC (option number136)", page 596
- Cycle 291 COUPLING TURNING INTERPOLATION (software option 96), was expanded by parameter Q561, see "COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)", page 303
- Cycles 421, 422 and 427 were expanded by the parameters Q498 and Q531, see "MEASURE HOLE (Cycle 421, DIN/ISO: G421)", page 545
- In cycle 247: SET DATUM, the datum point can be selected from the preset table, see "DATUM SETTING (Cycle 247, DIN/ISO: G247)", page 269
- In the cycles 200 and 203 the behavior of the dwell time was adjusted, see "UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)", page 86
- Cycle 205 performs deburring on the coordinate surface, see "UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)", page 92
- In SL cycles, M110 is now accounted for compensated inner arcs if activated during machining see "SL Cycles", page 196
New and changed cycle functions of software
Contents

1 Fundamentals / Overviews ................................................................. 51
2 Using Fixed Cycles ........................................................................ 55
3 Fixed Cycles: Drilling .................................................................... 75
4 Fixed Cycles: Tapping / Thread Milling ........................................ 105
5 Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling ............ 141
6 Fixed Cycles: Pattern Definitions .................................................. 185
7 Fixed Cycles: Contour Pocket ....................................................... 195
8 Fixed Cycles: Cylindrical Surface ................................................ 229
9 Fixed Cycles: Contour Pocket with Contour Formula .................. 247
10 Cycles: Coordinate Transformations .......................................... 261
11 Cycles: Special Functions ........................................................... 285
12 Cycles: Turning ........................................................................... 325
13 Using Touch Probe Cycles ............................................................ 447
14 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment .................................................. 457
15 Touch Probe Cycles: Automatic Datum Setting ............................ 479
16 Touch Probe Cycles: Automatic Workpiece Inspection .......... 533
17 Touch Probe Cycles: Special Functions ....................................... 579
18 Visual Setup Control VSC (software option 136) ..................... 595
19 Touch Probe Cycles: Automatic Kinematics Measurement ...... 617
20 Touch Probe Cycles: Automatic Tool Measurement ................ 649
21 Tables of Cycles ........................................................................ 665
1 Fundamentals / Overviews ..........................................................................................................................51

1.1 Introduction ..............................................................................................................................................52

1.2 Available Cycle Groups ..........................................................................................................................53

Overview of fixed cycles .................................................................................................................................53
Overview of touch probe cycles ......................................................................................................................54
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 Using Fixed Cycles</td>
<td>55</td>
</tr>
<tr>
<td>2.1 Working with fixed cycles</td>
<td>56</td>
</tr>
<tr>
<td>Machine-specific cycles</td>
<td>56</td>
</tr>
<tr>
<td>Defining a cycle using soft keys</td>
<td>57</td>
</tr>
<tr>
<td>Defining a cycle using the GOTO function</td>
<td>57</td>
</tr>
<tr>
<td>Calling a cycle</td>
<td>58</td>
</tr>
<tr>
<td>2.2 Program defaults for cycles</td>
<td>60</td>
</tr>
<tr>
<td>Overview</td>
<td>60</td>
</tr>
<tr>
<td>Entering GLOBAL DEF</td>
<td>60</td>
</tr>
<tr>
<td>Using GLOBAL DEF information</td>
<td>61</td>
</tr>
<tr>
<td>Global data valid everywhere</td>
<td>62</td>
</tr>
<tr>
<td>Global data for drilling operations</td>
<td>62</td>
</tr>
<tr>
<td>Global data for milling operations with pocket cycles 25x</td>
<td>62</td>
</tr>
<tr>
<td>Global data for milling operations with contour cycles</td>
<td>63</td>
</tr>
<tr>
<td>Global data for positioning behavior</td>
<td>63</td>
</tr>
<tr>
<td>Global data for probing functions</td>
<td>63</td>
</tr>
<tr>
<td>2.3 PATTERN DEF pattern definition</td>
<td>64</td>
</tr>
<tr>
<td>Application</td>
<td>64</td>
</tr>
<tr>
<td>Entering PATTERN DEF</td>
<td>65</td>
</tr>
<tr>
<td>Using PATTERN DEF</td>
<td>65</td>
</tr>
<tr>
<td>Defining individual machining positions</td>
<td>66</td>
</tr>
<tr>
<td>Defining a single row</td>
<td>66</td>
</tr>
<tr>
<td>Defining a single pattern</td>
<td>67</td>
</tr>
<tr>
<td>Defining individual frames</td>
<td>68</td>
</tr>
<tr>
<td>Defining a full circle</td>
<td>69</td>
</tr>
<tr>
<td>Defining a pitch circle</td>
<td>70</td>
</tr>
<tr>
<td>2.4 Point tables</td>
<td>71</td>
</tr>
<tr>
<td>Application</td>
<td>71</td>
</tr>
<tr>
<td>Creating a point table</td>
<td>71</td>
</tr>
<tr>
<td>Hiding single points from the machining process</td>
<td>72</td>
</tr>
<tr>
<td>Selecting a point table in the program</td>
<td>72</td>
</tr>
<tr>
<td>Calling a cycle in connection with point tables</td>
<td>73</td>
</tr>
</tbody>
</table>
3 Fixed Cycles: Drilling

3.1 Fundamentals

Overview

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

Cycle run

Please note while programming:

Cycle parameters

3.3 DRILLING (Cycle 200)

Cycle run

Please note while programming:

Cycle parameters

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

Cycle run

Please note while programming:

Cycle parameters

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

Cycle run

Please note while programming:

Cycle parameters

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

Cycle run

Please note while programming:

Cycle parameters

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

Cycle run

Please note while programming:

Cycle parameters

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

Cycle run

Please note while programming:

Cycle parameters
3.9 BORE MILLING (Cycle 208)................................................................................................................. 96
   Cycle run................................................................................................................................................. 96
   Please note while programming:........................................................................................................... 97
   Cycle parameters.................................................................................................................................... 98

3.10 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)................................................. 99
   Cycle run................................................................................................................................................. 99
   Please note while programming:........................................................................................................... 99
   Cycle parameters.................................................................................................................................. 100

3.11 Programming Examples..................................................................................................................... 102
   Example: Drilling cycles....................................................................................................................... 102
   Example: Using drilling cycles in connection with PATTERN DEF.................................................... 103
4 Fixed Cycles: Tapping / Thread Milling ................................................................. 105

4.1 Fundamentals ...................................................................................................... 106

Overview .................................................................................................................. 106

4.2 TAPPING with a floating tap holder (Cycle 206, DIN/ISO: G206) ...................... 107

Cycle run .................................................................................................................... 107

Please note while programming: .............................................................................. 108

Cycle parameters .................................................................................................... 109

4.3 RIGID TAPPING without a floating tap holder (Cycle 207, DIN/ISO: G207) .... 110

Cycle run .................................................................................................................... 110

Please note while programming: .............................................................................. 111

Cycle parameters .................................................................................................... 112

Retracting after a program interruption ................................................................. 112

4.4 TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209) ..................... 113

Cycle run .................................................................................................................... 113

Please note while programming: .............................................................................. 114

Cycle parameters .................................................................................................... 115

4.5 Fundamentals of Thread Milling ........................................................................ 117

Prerequisites ............................................................................................................. 117

4.6 THREAD MILLING (Cycle 262, DIN/ISO: G262) .............................................. 119

Cycle run .................................................................................................................... 119

Please note while programming: .............................................................................. 120

Cycle parameters .................................................................................................... 121

4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO:G263) ............ 122

Cycle run .................................................................................................................... 122

Please note while programming: .............................................................................. 123

Cycle parameters .................................................................................................... 124

4.8 THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264) ....................... 126

Cycle run .................................................................................................................... 126

Please note while programming: .............................................................................. 127

Cycle parameters .................................................................................................... 128
4.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265)........................................... 130

  Cycle run........................................................................................................................................... 130
  Please note while programming:........................................................................................................ 131
  Cycle parameters............................................................................................................................... 132

4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267).................................................... 134

  Cycle run........................................................................................................................................... 134
  Please note while programming:........................................................................................................ 135
  Cycle parameters............................................................................................................................... 136

4.11 Programming Examples................................................................................................................. 138

  Example: Thread milling..................................................................................................................... 138
## 5 Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>Fundamentals</td>
<td>142</td>
</tr>
<tr>
<td></td>
<td>Overview</td>
<td>142</td>
</tr>
<tr>
<td>5.2</td>
<td>RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251)</td>
<td>143</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>143</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>144</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>145</td>
</tr>
<tr>
<td>5.3</td>
<td>CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)</td>
<td>147</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>147</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>149</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>150</td>
</tr>
<tr>
<td>5.4</td>
<td>SLOT MILLING (Cycle 253, DIN/ISO: G253)</td>
<td>152</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>152</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>153</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>154</td>
</tr>
<tr>
<td>5.5</td>
<td>CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)</td>
<td>156</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>156</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>157</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>158</td>
</tr>
<tr>
<td>5.6</td>
<td>RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)</td>
<td>161</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>161</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>162</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>163</td>
</tr>
<tr>
<td>5.7</td>
<td>CIRCULAR STUD (Cycle 257, DIN/ISO: G257)</td>
<td>165</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>165</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>166</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>167</td>
</tr>
<tr>
<td>5.8</td>
<td>CIRCULAR STUD (cycle 258, DIN/ISO: G258)</td>
<td>169</td>
</tr>
<tr>
<td></td>
<td>Cycle run</td>
<td>169</td>
</tr>
<tr>
<td></td>
<td>Please note while programming</td>
<td>170</td>
</tr>
<tr>
<td></td>
<td>Cycle parameters</td>
<td>171</td>
</tr>
</tbody>
</table>
5.9 FACE MILLING (Cycle 233, DIN/ISO: G233) ................................................................. 174

- Cycle run .......................................................................................................................... 174
- Please note while programming: ...................................................................................... 178
- Cycle parameters ............................................................................................................. 179

5.10 Programming Examples ............................................................................................. 182

- Example: Milling pockets, studs and slots ........................................................................ 182
6 Fixed Cycles: Pattern Definitions ........................................................................................................ 185

6.1 Fundamentals ..................................................................................................................................... 186
    Overview............................................................................................................................................... 186

6.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220) .............................................................................. 187
    Cycle run............................................................................................................................................... 187
    Please note while programming: ......................................................................................................... 187
    Cycle parameters.................................................................................................................................. 188

6.3 LINEAR PATTERN (Cycle 221, DIN/ISO: G221) ............................................................................. 190
    Cycle run............................................................................................................................................... 190
    Please note while programming: ......................................................................................................... 190
    Cycle parameters.................................................................................................................................. 191

6.4 Programming Examples ..................................................................................................................... 192
    Example: Polar hole patterns ............................................................................................................... 192
7 Fixed Cycles: Contour Pocket

7.1 SL Cycles

Fundamentals
Overview

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

Please note while programming:
Cycle parameters

7.3 Superimposed contours

Fundamentals
Subprograms: overlapping pockets
Area of inclusion
Area of exclusion
Area of intersection

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

Please note while programming:
Cycle parameters

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

Cycle run
Please note while programming:
Cycle parameters

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

Cycle run
Please note while programming:
Cycle parameters

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

Cycle run
Please note while programming:
Cycle parameters

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

Cycle run
Please note while programming:
Cycle parameters
7.9 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125).................................................................216

   Cycle run.................................................................................................................................. 216
   Please note while programming:........................................................................................... 216
   Cycle parameters.................................................................................................................... 217

7.10 CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)..................................................218

   Please note while programming:........................................................................................... 218
   Cycle parameters.................................................................................................................... 218

7.11 TROCHOIDAL SLOT (Cycle 275, DIN ISO G275)............................................................219

   Cycle run.................................................................................................................................. 219
   Please note while programming:........................................................................................... 220
   Cycle parameters.................................................................................................................... 221

7.12 Programming Examples....................................................................................................223

   Example: Roughing-out and fine-roughing a pocket.............................................................. 223
   Example: Pilot drilling, roughing-out and finishing overlapping contours..............................225
   Example: Contour train........................................................................................................... 227
## 8 Fixed Cycles: Cylindrical Surface

8.1 Fundamentals................................................................. 230

Overview of cylindrical surface cycles........................................................... 230

8.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, software option 1)............................... 231

Cycle run................................................................................... 231
Please note while programming:............................................................ 232
Cycle parameters........................................................................... 233

8.3 CYLINDER SURFACE Slot milling (Cycle 28, DIN/ISO: G128, software option 1).............. 234

Cycle run................................................................................... 234
Please note while programming:............................................................ 235
Cycle parameters........................................................................... 236

8.4 CYLINDER SURFACE Ridge milling (Cycle 29, DIN/ISO: G129, software option 1)............ 237

Cycle run................................................................................... 237
Please note while programming:............................................................ 238
Cycle parameters........................................................................... 239

8.5 CYLINDER SURFACE (Cycle 39, DIN/ISO: G139, software option 1)............................... 240

Cycle run................................................................................... 240
Please note while programming:............................................................ 241
Cycle parameters........................................................................... 242

8.6 Programming Examples......................................................... 243

Example: Cylinder surface with Cycle 27........................................... 243
Example: Cylinder surface with Cycle 28........................................... 245
9 Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

Fundamentals
Selecting a program with contour definitions
Defining contour descriptions
Entering a complex contour formula
Superimposed contours
Contour machining with SL Cycles
Example: Roughing and finishing superimposed contours with the contour formula

9.2 SL cycles with simple contour formula

Fundamentals
Entering a simple contour formula
Contour machining with SL Cycles
## 10 Cycles: Coordinate Transformations

### 10.1 Fundamentals

- Overview
- Effect of coordinate transformations

### 10.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54)

- Effect
- Cycle parameters

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

- Effect
- Please note while programming
- Cycle parameters
- Selecting a datum table in the part program
- Edit the datum table in the Programming mode of operation
- Configuring the datum table
- To exit a datum table
- Status displays

### 10.4 DATUM SETTING (Cycle 247, DIN/ISO: G247)

- Effect
- Please note before programming
- Cycle parameters
- Status displays

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

- Effect
- Please note while programming
- Cycle parameters

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

- Effect
- Please note while programming
- Cycle parameters

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

- Effect
- Cycle parameters
10.8  AXIS-SPECIFIC SCALING (Cycle 26).................................................................................................................. 275

Effect........................................................................................................................................................................... 275
Please note while programming:...................................................................................................................................... 275
Cycle parameters......................................................................................................................................................... 276

10.9  WORKING PLANE (Cycle 19, DIN/ISO: G80, software option 1)........................................................................ 277

Effect........................................................................................................................................................................... 277
Please note while programming:...................................................................................................................................... 278
Cycle parameters......................................................................................................................................................... 278
Resetting........................................................................................................................................................................ 279
Positioning the axes of rotation...................................................................................................................................... 279
Position display in the tilted system.......................................................................................................................... 280
Workspace monitoring.................................................................................................................................................. 280
Positioning in a tilted coordinate system.................................................................................................................. 281
Combining coordinate transformation cycles............................................................................................................. 281
Procedure for working with Cycle 19 WORKING PLANE.......................................................................................... 282

10.10 Programming Examples..................................................................................................................................... 283

Example: Coordinate transformation cycles.................................................................................................................. 283
11 Cycles: Special Functions .................................................................................................................. 285

11.1 Fundamentals .................................................................................................................................. 286
Overview .................................................................................................................................................. 286

11.2 Dwell Time (Cycle 9, DIN/ISO: G04) ............................................................................................ 287
Function .................................................................................................................................................. 287
Cycle parameters .................................................................................................................................. 287

11.3 Program Call (Cycle 12, DIN/ISO: G39) ....................................................................................... 288
Cycle function ........................................................................................................................................ 288
Please note while programming: ........................................................................................................... 288
Cycle parameters .................................................................................................................................. 289

11.4 Spindle Orientation (Cycle 13, DIN/ISO: G36) ............................................................................. 290
Cycle function ........................................................................................................................................ 290
Please note while programming: ........................................................................................................... 290
Cycle parameters .................................................................................................................................. 290

11.5 Tolerance (Cycle 32, DIN/ISO: G62) ............................................................................................. 291
Cycle function ........................................................................................................................................ 291
Influences of the geometry definition in the CAM system ................................................................. 291
Please note while programming: ........................................................................................................... 292
Cycle parameters .................................................................................................................................. 293

11.6 Contour Turning Interpolation (Cycle 292, DIN/ISO: G292, software option 96) ............. 294
Cycle run ................................................................................................................................................ 294
Please note while programming: ........................................................................................................... 296
Cycle parameters .................................................................................................................................. 298
Machining variants .................................................................................................................................. 299
Defining the tool .................................................................................................................................... 301

11.7 Coupling Interpolation Turning (cycle 291, DIN/ISO: G291, software option 96) .......... 303
Cycle run ................................................................................................................................................ 303
Please note while programming: ........................................................................................................... 303
Cycle parameters .................................................................................................................................. 305
Defining the tool .................................................................................................................................... 306
11.8 ENGRAVING (Cycle 225, DIN/ISO: G225)................................................................. 309

Cycle run.......................................................................................................................... 309
Please note while programming:..................................................................................... 309
Cycle parameters.............................................................................................................. 310
Allowed engraving characters.......................................................................................... 311
Characters that cannot be printed................................................................................... 311
Engraving system variables.............................................................................................. 312

11.9 FACE MILLING (Cycle 232, DIN/ISO: G232)......................................................... 313

Cycle run.......................................................................................................................... 313
Please note while programming:..................................................................................... 315
Cycle parameters.............................................................................................................. 316

11.10 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 143)........ 318

Cycle run.......................................................................................................................... 318
Please note while programming:..................................................................................... 319
Cycle parameters.............................................................................................................. 319

11.11 Programming examples......................................................................................... 320

Example: Interpolation Turning Cycle 291................................................................. 320
Example: Interpolation Turning Cycle 292................................................................. 322
12 Cycles: Turning

12.1 Turning Cycles (software option 50)

Overview .......................................................................................................................... 326
Working with turning cycles ......................................................................................... 329
Blank form update (FUNCTION TURNDATA) ............................................................ 330

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

Application .................................................................................................................... 332
Effect .............................................................................................................................. 335
Please note while programming ................................................................................... 335
Cycle parameters ......................................................................................................... 336

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

Please note while programming ................................................................................... 338
Effect .............................................................................................................................. 338
Cycle parameters ......................................................................................................... 338

12.4 Fundamentals of Turning Cycles

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

Application .................................................................................................................... 340
Roughing cycle run ....................................................................................................... 340
Finishing cycle run ....................................................................................................... 341
Please note while programming ................................................................................... 341
Cycle parameters ......................................................................................................... 342

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

Application .................................................................................................................... 343
Roughing cycle run ....................................................................................................... 343
Finishing cycle run ....................................................................................................... 344
Please note while programming ................................................................................... 344
Cycle parameters ......................................................................................................... 345

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

Application .................................................................................................................... 347
Roughing cycle run ....................................................................................................... 347
Finishing cycle run ....................................................................................................... 348
Please note while programming ................................................................................... 348
Cycle parameters ......................................................................................................... 349
12.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814) ........................................... 350

Application............................................................................................................................................. 350
Roughing cycle run................................................................................................................................. 350
Finishing cycle run................................................................................................................................. 351
Please note while programming:......................................................................................................351
Cycle parameters.................................................................................................................................. 352

12.9 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810) .................................................... 354

Application............................................................................................................................................. 354
Roughing cycle run................................................................................................................................. 354
Finishing cycle run................................................................................................................................. 355
Please note while programming:......................................................................................................355
Cycle parameters.................................................................................................................................. 356

12.10 TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815) ............................................................. 358

Application............................................................................................................................................. 358
Roughing cycle run................................................................................................................................. 358
Finishing cycle run................................................................................................................................. 359
Please note while programming:......................................................................................................359
Cycle parameters.................................................................................................................................. 360

12.11 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821) ................................................................. 362

Application............................................................................................................................................. 362
Roughing cycle run................................................................................................................................. 362
Finishing cycle run................................................................................................................................. 363
Please note while programming:......................................................................................................363
Cycle parameters.................................................................................................................................. 364

12.12 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822) .......................................... 365

Application............................................................................................................................................. 365
Roughing cycle run................................................................................................................................. 365
Finishing cycle run................................................................................................................................. 366
Please note while programming:......................................................................................................366
Cycle parameters.................................................................................................................................. 367
12.13 TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823) ............................................................... 369
Application............................................................................................................................................. 369
Roughing cycle run ................................................................................................................................. 369
Finishing cycle run ................................................................................................................................. 370
Please note while programming: ........................................................................................................ 370
Cycle parameters.................................................................................................................................. 371

12.14 TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824) ...................................... 372
Application............................................................................................................................................. 372
Roughing cycle run ................................................................................................................................. 372
Finishing cycle run ................................................................................................................................. 373
Please note while programming: ........................................................................................................ 373
Cycle parameters.................................................................................................................................. 374

12.15 TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820) .................................................................... 376
Application............................................................................................................................................. 376
Roughing cycle run ................................................................................................................................. 376
Finishing cycle run ................................................................................................................................. 377
Please note while programming: ........................................................................................................ 377
Cycle parameters.................................................................................................................................. 378

12.16 SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841) .......................................................... 380
Application............................................................................................................................................. 380
Roughing cycle run ................................................................................................................................. 380
Finishing cycle run ................................................................................................................................. 381
Please note while programming: ........................................................................................................ 381
Cycle parameters.................................................................................................................................. 382

12.17 RADIAL RECESSING EXTENDED (Cycle 842, DIN/ISO: G842) ...................................................... 383
Application............................................................................................................................................. 383
Roughing cycle run ................................................................................................................................. 383
Finishing cycle run ................................................................................................................................. 384
Please note while programming: ........................................................................................................ 384
Cycle parameters.................................................................................................................................. 385
12.18 RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840) ................................................................. 388

Application ................................................................................................................................................. 388
Roughing cycle run ................................................................................................................................. 388
Finishing cycle run .............................................................................................................................. 389
Please note while programming: .......................................................................................................... 389
Cycle parameters .................................................................................................................................. 390

12.19 SIMPLE AXIAL RECESSING (Cycle 851, DIN/ISO: G851) ................................................................. 392

Application ................................................................................................................................................. 392
Roughing cycle run ................................................................................................................................. 392
Finishing cycle run ............................................................................................................................... 393
Please note while programming: .......................................................................................................... 393
Cycle parameters .................................................................................................................................. 394

12.20 AXIAL RECESSING EXTENDED (Cycle 852, DIN/ISO: G852) ............................................................ 395

Application ................................................................................................................................................. 395
Roughing cycle run ................................................................................................................................. 395
Finishing cycle run ............................................................................................................................... 396
Please note while programming: .......................................................................................................... 396
Cycle parameters .................................................................................................................................. 397

12.21 AXIAL RECESSING (Cycle 850, DIN/ISO: G850) .................................................................................. 400

Application ................................................................................................................................................. 400
Roughing cycle run ................................................................................................................................. 400
Finishing cycle run ............................................................................................................................... 401
Please note while programming: .......................................................................................................... 401
Cycle parameters .................................................................................................................................. 402

12.22 RADIAL RECESSING (Cycle 861, DIN/ISO: G861) .............................................................................. 404

Application ................................................................................................................................................. 404
Roughing cycle run ................................................................................................................................. 404
Finishing cycle run ............................................................................................................................... 405
Please note while programming: .......................................................................................................... 405
Cycle parameters .................................................................................................................................. 406
## Contents

### 12.23 Radial Recessing Extended (Cycle 862, DIN/ISO: G862)

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Application</td>
<td>407</td>
</tr>
<tr>
<td>Roughing cycle run</td>
<td>407</td>
</tr>
<tr>
<td>Finishing cycle run</td>
<td>408</td>
</tr>
<tr>
<td>Please note while programming:</td>
<td>408</td>
</tr>
<tr>
<td>Cycle parameters</td>
<td>409</td>
</tr>
</tbody>
</table>

### 12.24 Recessing Contour Radial (Cycle 860, DIN/ISO: G860)

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Application</td>
<td>411</td>
</tr>
<tr>
<td>Roughing cycle run</td>
<td>411</td>
</tr>
<tr>
<td>Finishing cycle run</td>
<td>412</td>
</tr>
<tr>
<td>Please note while programming:</td>
<td>412</td>
</tr>
<tr>
<td>Cycle parameters</td>
<td>413</td>
</tr>
</tbody>
</table>

### 12.25 Axial Recessing (Cycle 871, DIN/ISO: G871)

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Application</td>
<td>415</td>
</tr>
<tr>
<td>Roughing cycle run</td>
<td>415</td>
</tr>
<tr>
<td>Finishing cycle run</td>
<td>415</td>
</tr>
<tr>
<td>Please note while programming:</td>
<td>416</td>
</tr>
<tr>
<td>Cycle parameters</td>
<td>416</td>
</tr>
</tbody>
</table>

### 12.26 Axial Recessing Extended (Cycle 872, DIN/ISO: G872)

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Application</td>
<td>417</td>
</tr>
<tr>
<td>Roughing cycle run</td>
<td>417</td>
</tr>
<tr>
<td>Finishing cycle run</td>
<td>418</td>
</tr>
<tr>
<td>Please note while programming:</td>
<td>418</td>
</tr>
<tr>
<td>Cycle parameters</td>
<td>419</td>
</tr>
</tbody>
</table>

### 12.27 Axial Recessing (Cycle 870, DIN/ISO: G870)

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Application</td>
<td>421</td>
</tr>
<tr>
<td>Roughing cycle run</td>
<td>421</td>
</tr>
<tr>
<td>Finishing cycle run</td>
<td>422</td>
</tr>
<tr>
<td>Please note while programming:</td>
<td>422</td>
</tr>
<tr>
<td>Cycle parameters</td>
<td>423</td>
</tr>
</tbody>
</table>
12.28 THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831) ......................................................... 424

   Application..................................................................................................................................... 424
   Cycle run....................................................................................................................................... 424
   Please note while programming:................................................................................................. 425
   Cycle parameters........................................................................................................................... 426

12.29 THREAD EXTENDED (Cycle 832, DIN/ISO: G832) ............................................................. 427

   Application..................................................................................................................................... 427
   Cycle run....................................................................................................................................... 427
   Please note while programming:................................................................................................. 428
   Cycle parameters........................................................................................................................... 429

12.30 CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ISO: G830) .............................................. 431

   Application..................................................................................................................................... 431
   Cycle run....................................................................................................................................... 431
   Please note while programming:................................................................................................. 432
   Cycle parameters........................................................................................................................... 433

12.31 GEAR HOBBING (Cycle 880, DIN/ISO: G880) ..................................................................... 435

   Cycle run....................................................................................................................................... 435
   Please note while programming:................................................................................................. 436
   Cycle parameters........................................................................................................................... 437
   Direction of rotation depending on the machining side (Q550)..................................................... 439

12.32 CHECK UNBALANCE (Cycle 892, DIN/ISO: G892) ............................................................ 440

   Application..................................................................................................................................... 440
   Please note while programming:................................................................................................. 441
   Cycle parameters........................................................................................................................... 442

12.33 Example program...................................................................................................................... 443

   Example: Shoulder with recess.................................................................................................... 443
   Example: Gear hobbing................................................................................................................. 445
13 Using Touch Probe Cycles........................................................................................................... 447

13.1 General information about touch probe cycles................................................................. 448

Method of function.............................................................................................................................. 448
Consideration of a basic rotation in the Manual Operation mode.................................................. 448
Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes........... 448
Touch probe cycles for automatic operation.................................................................................. 449

13.2 Before You Start Working with Touch Probe Cycles....................................................... 451

Maximum traverse to touch point: DIST in touch probe table.................................................... 451
Set-up clearance to touch point: SET_UP in touch probe table.................................................... 451
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table.. 451
Touch trigger probe, probing feed rate: F in touch probe table.................................................... 452
Touch trigger probe, rapid traverse for positioning: FMAX.......................................................... 452
Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table................. 452
Multiple measurements.................................................................................................................. 453
Confidence interval of multiple measurements............................................................................. 453
Executing touch probe cycles......................................................................................................... 454

13.3 Touch probe table.................................................................................................................. 455

General information.......................................................................................................................... 455
Editing touch probe tables.............................................................................................................. 455
Touch probe data.............................................................................................................................. 456
14 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.1 Fundamentals

Overview

Characteristics common to all touch probe cycles for measuring workpiece misalignment

14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

Cycle run

Please note while programming

Cycle parameters

14.3 BASIC ROTATION over two holes (Cycle 401, DIN/ISO: G401)

Cycle run

Please note while programming

Cycle parameters

14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402)

Cycle run

Please note while programming

Cycle parameters

14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403)

Cycle run

Please note while programming

Cycle parameters

14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)

Cycle run

Cycle parameters

14.7 Compensating workpiece misalignment by rotating the C axis (Cycle 405, DIN/ISO: G405)

Cycle run

Please note while programming

Cycle parameters

14.8 Example: Determining a basic rotation from two holes
15 Touch Probe Cycles: Automatic Datum Setting ................................................................. 479

15.1 Fundamentals ....................................................................................................................... 480

Overview ........................................................................................................................................ 480
Characteristics common to all touch probe cycles for datum setting.............................................. 482

15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408) .......................................................... 484

Cycle run ......................................................................................................................................... 484
Please note while programming: ........................................................................................................ 485
Cycle parameters ............................................................................................................................. 486

15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409) ......................................................... 488

Cycle run ......................................................................................................................................... 488
Please note while programming: ........................................................................................................ 488
Cycle parameters ............................................................................................................................. 489

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410) ................................. 491

Cycle run ......................................................................................................................................... 491
Please note while programming: ........................................................................................................ 492
Cycle parameters ............................................................................................................................. 493

15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411) ......................... 495

Cycle run ......................................................................................................................................... 495
Please note while programming: ........................................................................................................ 495
Cycle parameters ............................................................................................................................. 496

15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412) ....................................... 498

Cycle run ......................................................................................................................................... 498
Please note while programming: ........................................................................................................ 499
Cycle parameters ............................................................................................................................. 500

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413) ..................................... 503

Cycle run ......................................................................................................................................... 503
Please note while programming: ........................................................................................................ 503
Cycle parameters ............................................................................................................................. 504

15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414) ................................. 507

Cycle run ......................................................................................................................................... 507
Please note while programming: ........................................................................................................ 508
Cycle parameters ............................................................................................................................. 509
15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415)......................................................... 512
   Cycle run............................................................................................................................................. 512
   Please note while programming:........................................................................................................ 513
   Cycle parameters................................................................................................................................ 514

15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)................................................................. 516
   Cycle run............................................................................................................................................. 516
   Please note while programming:........................................................................................................ 517
   Cycle parameters................................................................................................................................ 518

15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417)..................................................... 520
   Cycle run............................................................................................................................................. 520
   Please note while programming:........................................................................................................ 520
   Cycle parameters................................................................................................................................ 521

15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)...................................................... 522
   Cycle run............................................................................................................................................. 522
   Please note while programming:........................................................................................................ 523
   Cycle parameters................................................................................................................................ 524

15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)................................................................. 526
   Cycle run............................................................................................................................................. 526
   Please note while programming:........................................................................................................ 526
   Cycle parameters................................................................................................................................ 527

15.14 Example: Datum setting in center of a circular segment and on top surface of workpiece......... 529

15.15 Example: Datum setting on top surface of workpiece and in center of a bolt hole circle......... 530
## 16 Touch Probe Cycles: Automatic Workpiece Inspection

### 16.1 Fundamentals

- Overview
- Recording the results of measurement
- Measurement results in Q parameters
- Classification of results
- Tolerance monitoring
- Tool monitoring
- Reference system for measurement results

### 16.2 DATUM PLANE (Cycle 0, DIN/ISO: G55)

- Cycle run
- Please note while programming
- Cycle parameters

### 16.3 POLAR DATUM PLANE (Cycle 1)

- Cycle run
- Please note while programming
- Cycle parameters

### 16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

- Cycle run
- Please note while programming
- Cycle parameters

### 16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)

- Cycle run
- Please note while programming
- Cycle parameters

### 16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

- Cycle run
- Please note while programming
- Cycle parameters

### 16.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423)

- Cycle run
- Please note while programming
- Cycle parameters
16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424) ...................................................... 558

Cycle run............................................................................................................................................... 558
Please note while programming:........................................................................................................... 558
Cycle parameters.................................................................................................................................. 559

16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425) .................................................................. 561

Cycle run............................................................................................................................................... 561
Please note while programming:........................................................................................................... 561
Cycle parameters.................................................................................................................................. 562

16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426) ........................................................... 564

Cycle run............................................................................................................................................... 564
Please note while programming:........................................................................................................... 564
Cycle parameters.................................................................................................................................. 565

16.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427) ............................................................ 567

Cycle run............................................................................................................................................... 567
Please note while programming:........................................................................................................... 567
Cycle parameters.................................................................................................................................. 568

16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430) ................................................... 570

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

16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431) ................................................................. 573

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

16.14 Programming Examples.............................................................................................................. 576

Example: Measuring and reworking a rectangular stud...................................................................... 576
Example: Measuring a rectangular pocket and recording the results...................................................... 578
## 17 Touch Probe Cycles: Special Functions

### 17.1 Fundamentals

Overview

### 17.2 MEASURE (Cycle 3)

Cycle run

Please note while programming:

Cycle parameters

### 17.3 MEASURING IN 3-D (Cycle 4)

Cycle run

Please note while programming:

Cycle parameters

### 17.4 Calibrating a touch trigger probe

### 17.5 Displaying calibration values

### 17.6 CALIBRATE TS (Cycle 460, DIN/ISO: G460)

### 17.7 CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461)

### 17.8 CALIBRATE TS RADIUS INSIDE (Cycle 462, DIN/ISO: G462)

### 17.9 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, DIN/ISO: G463)
18 Visual Setup Control VSC (software option 136)........................................................................595

18.1 Camera-based monitoring of the setup situation VSC (option number 136)..........................596

- Fundamentals..................................................................................................................................596
- Produce live image.............................................................................................................................598
- Manage monitoring data....................................................................................................................599
- Overview.........................................................................................................................................601
- Results of the image evaluation........................................................................................................602
- Configuration....................................................................................................................................602
- Defining the monitoring area............................................................................................................604
- Possible queries.................................................................................................................................605

18.2 Workspace Global (Cycle 600). ...............................................................................................606

- Application.......................................................................................................................................606
- Creating reference images................................................................................................................606
- Monitoring phase.............................................................................................................................609
- Please note while programming:.....................................................................................................609
- Cycle parameters..............................................................................................................................610

18.3 Workspace Local (Cycle 601). .................................................................................................611

- Application.......................................................................................................................................611
- Creating reference images................................................................................................................611
- Monitoring phase.............................................................................................................................614
- Please note while programming:.....................................................................................................614
- Cycle parameters..............................................................................................................................615
## 19 Touch Probe Cycles: Automatic Kinematics Measurement

### 19.1 Kinematics Measurement with TS Touch Probes (KinematicsOpt option)

- **Fundamentals**
- **Overview**

### 19.2 Prerequisites

Please note while programming:

### 19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option)

- **Cycle run**
- **Please note while programming:**
- **Cycle parameters**
- **Logging function**
- **Notes on data management**

### 19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

- **Cycle run**
- **Positioning direction**
- **Machines with Hirth-coupled axes**
- **Choice of number of measuring points**
- **Choice of the calibration sphere position on the machine table**
- **Notes on the accuracy**
- **Notes on various calibration methods**
- **Backlash**
- **Please note while programming:**
- **Cycle parameters**
- **Various modes (Q406)**
- **Logging function**

### 19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- **Cycle run**
- **Please note while programming:**
- **Cycle parameters**
- **Adjustment of interchangeable heads**
- **Drift compensation**
- **Logging function**
20 Touch Probe Cycles: Automatic Tool Measurement

20.1 Fundamentals

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

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

Cycle run
Please note while programming
Cycle parameters

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

Fundamentals
Cycle run
Please note while programming
Cycle parameters

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

Cycle run
Please note while programming
Cycle parameters

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

Cycle run
Please note while programming
Cycle parameters

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

Cycle run
Please note while programming
Cycle parameters
21 Tables of Cycles ................................................................. 665

21.1 Overview ........................................................................ 666

- Fixed cycles .................................................................... 666
- Turning cycles ................................................................ 668
- Touch probe cycles ......................................................... 669
1.1 Introduction

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

Danger of collision!

Cycles sometimes execute extensive operations. For safety reasons, you should run a graphical program test before machining.

If you use indirect parameter assignments in cycles with numbers greater than 200 (e.g. Q210 = Q1), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. Q210) directly in such cases.

If you define a feed-rate parameter for fixed cycles greater than 200, then instead of entering a numerical value you can use soft keys to assign the feed rate defined in the TOOL CALL block (FAUTO soft key). You can also use the feed-rate alternatives FMAX (rapid traverse), FZ (feed per tooth) and FU (feed per rev), depending on the respective cycle and the function of the feed-rate parameter.

Note that, after a cycle definition, a change of the FAUTO feed rate has no effect, because internally the TNC assigns the feed rate from the TOOL CALL block when processing the cycle definition.

If you want to delete a block that is part of a cycle, the TNC asks you whether you want to delete the whole cycle.
1.2 **Available Cycle Groups**

**Overview of fixed cycles**

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

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle group</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>DRILLING/THREAD</td>
<td>Cycles for pecking, reaming, boring and counterboring</td>
<td>76</td>
</tr>
<tr>
<td>DRILLING/THREAD</td>
<td>Cycles for tapping, thread cutting and thread milling</td>
<td>106</td>
</tr>
<tr>
<td>POCKETS/STUDS/SLOTS</td>
<td>Cycles for milling pockets, studs and slots and for face milling</td>
<td>142</td>
</tr>
<tr>
<td>COORD. TRANSF.</td>
<td>Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours</td>
<td>262</td>
</tr>
<tr>
<td>SL CYCLES</td>
<td>Subcontour List (SL) cycles, which allow the machining of contours consisting of several overlapping subcontours, as well as cycles for cylinder surface machining and for trochoidal milling</td>
<td>230</td>
</tr>
<tr>
<td>PATTERN</td>
<td>Cycles for producing point patterns, such as circular or linear hole patterns</td>
<td>186</td>
</tr>
<tr>
<td>TURNING</td>
<td>Cycles for turning and gear hobbing</td>
<td>326</td>
</tr>
<tr>
<td>SPECIAL CYCLES</td>
<td>Special cycles such as dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, ascertaining the load</td>
<td>286</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>Soft key</th>
<th>Cycle group</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cycles for automatic measurement and compensation of workpiece misalignment</td>
<td>458</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic workpiece presetting</td>
<td>480</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic workpiece inspection</td>
<td>534</td>
</tr>
<tr>
<td></td>
<td>Special cycles</td>
<td>580</td>
</tr>
<tr>
<td></td>
<td>Touch probe calibration</td>
<td>587</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic kinematics measurement</td>
<td>458</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic tool measurement (enabled by the machine tool builder)</td>
<td>650</td>
</tr>
<tr>
<td></td>
<td>Cycles for Visual Setup Control VSC (software option 136)</td>
<td>596</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

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

Sometimes machine-specific cycles use transfer parameters that HEIDENHAIN already uses in standard cycles. For parallel use of DEF active cycles (cycles that the TNC is automatically running during cycle definition) and CALL active cycles (cycles that you need to call up to run).

Further Information: Calling a cycle, page 58
Adhere to the following procedure in order to avoid problems regarding the overwriting of transfer parameters that are used more than once:
- As a rule, always program DEF-active cycles before CALL-active cycles
- If you do want to program a DEF-active cycle between the definition and call of a CALL-active cycle, do it only if there is no common use of specific transfer parameters
Defining a cycle using soft keys

- The soft-key row shows the available groups of cycles
- Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles
- Select the cycle, e.g. THREAD MILLING. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the \texttt{ENT} key
- The TNC ends the dialog when all required data has been entered

Defining a cycle using the GOTO function

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

Example NC blocks

<table>
<thead>
<tr>
<th></th>
<th>CYCL DEF 200 DRILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2</td>
<td>;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q201=3</td>
<td>;DEPTH</td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PLNGNG</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 SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q211=0.25</td>
<td>;DWELL TIME AT DEPTH</td>
</tr>
<tr>
<td>Q395=0</td>
<td>;DEPTH REFERENCE</td>
</tr>
</tbody>
</table>
Using Fixed Cycles
2.1 Working with fixed cycles

Calling a cycle

Prerequisites
The following data must always be programmed before a cycle call:
- BLK FORM for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF)

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

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:
- Cycle 220 for point patterns on circles and Cycle 221 for point patterns on lines
- SL Cycle 14 CONTOUR GEOMETRY
- SL Cycle 20 CONTOUR DATA
- Cycle 32 TOLERANCE
- Coordinate transformation cycles
- Cycle 9 DWELL TIME
- All touch probe cycles

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

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

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

Calling a cycle with CYCL CALL PAT
The CYCL CALL PAT function calls the most recently defined machining cycle at all positions that you defined in a PATTERN DEF pattern definition or in a points table.

Further Information: PATTERN DEF pattern definition, page 64
Further Information: Point tables, page 71
Calling a cycle with CYCL CALL POS

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

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

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

Three coordinate axes must always be programmed in the CYCL CALL POS block. With the coordinate in the tool axis you can easily change the starting position. It serves as an additional datum shift.

The feed rate most recently defined in the CYCL CALL POS block applies only to traverse to the start position programmed in this block.

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

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

Cycle call with M99/M89

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

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

To cancel the effect of M89, program:

- M99 in the positioning block in which you move to the last starting point, or
- Use CYCL DEF to define a new fixed cycle
2.2 Program defaults for cycles

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

The following GLOBAL DEF functions are available:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Machining patterns</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>GLOBAL DEF COMMON</td>
<td>62</td>
</tr>
<tr>
<td></td>
<td>Definition of generally valid cycle parameters</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>GLOBAL DEF DRILLING</td>
<td>62</td>
</tr>
<tr>
<td></td>
<td>Definition of specific drilling cycle parameters</td>
<td></td>
</tr>
<tr>
<td>111</td>
<td>GLOBAL DEF POCKET MILLING</td>
<td>62</td>
</tr>
<tr>
<td></td>
<td>Definition of specific pocket-milling cycle parameters</td>
<td></td>
</tr>
<tr>
<td>111</td>
<td>GLOBAL DEF CONTOUR MILLING</td>
<td>63</td>
</tr>
<tr>
<td></td>
<td>Definition of specific contour milling cycle parameters</td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>GLOBAL DEF POSITIONING</td>
<td>63</td>
</tr>
<tr>
<td></td>
<td>Definition of the positioning behavior for CYCL CALL PAT</td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>GLOBAL DEF PROBING</td>
<td>63</td>
</tr>
<tr>
<td></td>
<td>Definition of specific touch probe cycle parameters</td>
<td></td>
</tr>
</tbody>
</table>

Entering GLOBAL DEF

- Select the Programming mode of operation
- Press the special functions key
- Select the functions for program defaults
- Select GLOBAL DEF functions
- Select desired GLOBAL DEF function, e.g. GLOBAL DEF GENERAL
- Enter the required definitions, and confirm each entry with the key ENT
Using GLOBAL DEF information

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

Proceed as follows:

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

Danger of collision!

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

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

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

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

Global data for drilling operations

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

The parameters apply to the drilling, tapping and thread milling cycles 200 to 209, 240, 241 and 262 to 267.

Global data for milling operations with pocket cycles 25x

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

The parameters apply to milling cycles 251 to 257.
Global data for milling operations with contour cycles

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

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

Global data for positioning behavior

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

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

Global data for probing functions

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

Applies to all Touch Probe Cycles 4xx.
2.3 PATTERN DEF pattern definition

Application

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

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

The following machining patterns are available:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Machining patterns</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>POINT</td>
<td>Definition of up to any 9 machining positions</td>
<td>66</td>
</tr>
<tr>
<td>ROW</td>
<td>Definition of a single row, straight or rotated</td>
<td>66</td>
</tr>
<tr>
<td>PATTERN</td>
<td>Definition of a single pattern, straight, rotated or distorted</td>
<td>67</td>
</tr>
<tr>
<td>FRAME</td>
<td>Definition of a single frame, straight, rotated or distorted</td>
<td>68</td>
</tr>
<tr>
<td>CIRCLE</td>
<td>Definition of a full circle</td>
<td>69</td>
</tr>
<tr>
<td>PITCH CIRCLE</td>
<td>Definition of a pitch circle</td>
<td>70</td>
</tr>
</tbody>
</table>
Entering PATTERN DEF

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

Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the CYCL CALL PAT function.

Further Information: Calling a cycle, page 58

The TNC then performs the most recently defined machining cycle on the machining pattern you defined.

A machining pattern remains active until you define a new one, or select a point table with the SEL PATTERN function.
You can use the mid-program startup function to select any point at which you want to start or continue machining.

More information: User’s Manual for conversational programming

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

2.3 PATTERN DEF pattern definition

Defining individual machining positions

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

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

NC blocks

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

Defining a single row

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

- Starting point in X (absolute): Coordinate of the starting point of the row in the X axis
- Starting point in Y (absolute): Coordinate of the starting point of the row in the Y axis
- Spacing of machining positions (incremental): Distance between the machining positions. You can enter a positive or negative value
- Number of repetitions: Total number of machining operations
- Rot. position of entire pattern (absolute): Angle of rotation around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z1). You can enter a positive or negative value
- Coordinate of workpiece surface (absolute): Enter Z coordinate at which machining is to begin

NC blocks

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

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

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

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

**NC blocks**

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

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

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

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

---

**NC blocks**

<table>
<thead>
<tr>
<th>Number</th>
<th>Command</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 L Z+100 R0 FMAX</td>
<td>(X+25\ Y+33.5\ \text{DX}+8\ \text{DY}+10\ \text{NUMX5}\ \text{NUMY4\ ROT}+0\ \text{ROTX}+0\ \text{ROTY}+0\ \text{Z}+0)</td>
<td>(X+25\ Y+33.5\ \text{DX}+8\ \text{DY}+10\ \text{NUMX5}\ \text{NUMY4\ ROT}+0\ \text{ROTX}+0\ \text{ROTY}+0\ \text{Z}+0)</td>
</tr>
</tbody>
</table>
Defining a full circle

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

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

**NC blocks**

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

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

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 L Z+100 R0 FMAX</td>
</tr>
<tr>
<td>11 PATTERN DEF PITCHCIRC1 (X+25 Y+33 D80 START+45 STEP30 NUM8 Z+0)</td>
</tr>
</tbody>
</table>
2.4 Point tables

Application
You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern. If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table
- Select the Programming mode of operation
- Call the file manager: Press the PGM MGT key.

FILE NAME?
- Enter the name and file type of the point table and confirm your entry with the ENT key.
- Select the unit of measure: Press the MM or INCH soft key. The TNC changes to the program blocks window and displays an empty point table.
- With the INSERT LINE soft key, insert new lines and enter the coordinates of the desired machining position.

Repeat the process until all desired coordinates have been entered.

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

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

- In the table, select the point to be hidden
- Select the FADE column
- Activate hiding, or
- Deactivate hiding

Selecting a point table in the program

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

- Press the PGM CALL key to call the function for selecting the point table
- Press the POINT TABLE soft key

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

Example NC block

7 SEL PATTERN "TNC:DIRKT5NUST35.PNT"
Calling a cycle in connection with point tables

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

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

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

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

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

Effect of the point table with SL cycles and Cycle 12

The TNC interprets the points as an additional datum shift.

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

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

Effect of the point table with Cycles 251 to 254

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

Overview

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

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>740</td>
<td>240 CENTERING</td>
<td>With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth</td>
<td>77</td>
</tr>
<tr>
<td>740</td>
<td>200 DRILLING</td>
<td>With automatic pre-positioning, 2nd set-up clearance</td>
<td>79</td>
</tr>
<tr>
<td>741</td>
<td>201 REAMING</td>
<td>With automatic pre-positioning, 2nd set-up clearance</td>
<td>81</td>
</tr>
<tr>
<td>742</td>
<td>202 BORING</td>
<td>With automatic pre-positioning, 2nd set-up clearance</td>
<td>83</td>
</tr>
<tr>
<td>742</td>
<td>203 UNIVERSAL DRILLING</td>
<td>With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing</td>
<td>86</td>
</tr>
<tr>
<td>742</td>
<td>204 BACK BORING</td>
<td>With automatic pre-positioning, 2nd set-up clearance</td>
<td>89</td>
</tr>
<tr>
<td>744</td>
<td>205 UNIVERSAL PECKING</td>
<td>With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance</td>
<td>92</td>
</tr>
<tr>
<td>746</td>
<td>208 BORE MILLING</td>
<td>With automatic pre-positioning, 2nd set-up clearance</td>
<td>96</td>
</tr>
<tr>
<td>746</td>
<td>241 SINGLE-LIP D.H.DRLNG</td>
<td>With automatic pre-positioning to deepened starting point, shaft speed and coolant definition</td>
<td>99</td>
</tr>
</tbody>
</table>
3.2 CENTERING (Cycle 240, DIN/ISO: G240)

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

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 $Q_{344}$ (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!
- Use the machine parameter $\text{displayDepthErr}$ to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).
- Keep in mind that the TNC reverses the calculation for pre-positioning when a positive diameter or depth is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!
Fixed Cycles: Drilling

3.2 CENTERING (Cycle 240)

**Cycle parameters**

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

**NC blocks**

<table>
<thead>
<tr>
<th>NC block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 L Z+100 R0 FMAX</td>
<td></td>
</tr>
<tr>
<td>11 CYCL DEF 240 CENTERING</td>
<td></td>
</tr>
<tr>
<td>Q200=2</td>
<td>;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q343=1</td>
<td>;SELECT DIA./DEPTH</td>
</tr>
<tr>
<td>Q201=+0</td>
<td>;DEPTH</td>
</tr>
<tr>
<td>Q344=-9</td>
<td>;DIAMETER</td>
</tr>
<tr>
<td>Q206=250</td>
<td>;FEED RATE FOR PLUNGING</td>
</tr>
<tr>
<td>Q211=0.1</td>
<td>;Dwell time at depth</td>
</tr>
<tr>
<td>Q203=+20</td>
<td>;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=100</td>
<td>;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>12 L X+30 Y+20 R0 FMAX M3 M99</td>
<td></td>
</tr>
<tr>
<td>13 L X+80 Y+50 R0 FMAX M99</td>
<td></td>
</tr>
</tbody>
</table>
3.3 DRILLING (Cycle 200)

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

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 \( \text{DEPTH} \) determines the working direction. If you program \( \text{DEPTH}=0 \), the cycle will not be executed.
- **Danger of collision!** Use the machine parameter \( \text{displayDepthErr} \) to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off). Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!
Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.9999, 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 set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Depth reference** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the tip angle of the tool must be defined in the T ANGLE column of the TOOL.T tool table.
  - 0 = depth in relation to the tool tip
  - 1 = depth in relation to the cylindrical part of the tool

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>11 CYCL DEF 200 DRILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q201=-15 ;DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q206=250 ;FEED RATE FOR PLUNGING</td>
<td></td>
</tr>
<tr>
<td>Q202=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q211=0 ;DWELL TIME AT TOP</td>
<td></td>
</tr>
<tr>
<td>Q203=+20 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>Q204=100 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q211=0.1 ;DWELL TIME AT BOTTOM</td>
<td></td>
</tr>
<tr>
<td>Q395=0 ;DEPTH REFERENCE</td>
<td></td>
</tr>
<tr>
<td>12 L X+30 Y+20 FMAX M3</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL</td>
<td></td>
</tr>
<tr>
<td>14 L X+80 Y+50 FMAX M99</td>
<td></td>
</tr>
</tbody>
</table>
3.4 REAMING (Cycle 201, DIN/ISO: G201)

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

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!
  - Use the machine parameter `displayDepthErr` to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).
  - Keep in mind that the TNC reverses the calculation for pre-positioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!
3.4 REAMING (Cycle 201)

Cycle parameters

- **Set-up clearance** $Q_{200}$ (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** $Q_{201}$ (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** $Q_{206}$: Traversing speed of the tool during reaming in mm/min. Input range 0 to 99999.9999; alternatively $FAUTO$, $FU$
- **Dwell time at depth** $Q_{211}$: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Retraction feed rate** $Q_{208}$: Traversing speed of the tool in mm/min when retracting from the hole. If you enter $Q_{208} = 0$, the tool retracts at the reaming feed rate. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** $Q_{203}$ (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999
- **2nd set-up clearance** $Q_{204}$ (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

NC blocks

```
11 CYCL DEF 201 REAMING
Q200=2 ;SET-UP CLEARANCE
Q201=-15 ;DEPTH
Q206=100 ;FEED RATE FOR PLUNGNG
Q211=0.5 ;DWELL TIME AT BOTTOM
Q208=250 ;RETRACTION FEED RATE
Q203=+20 ;SURFACE COORDINATE
Q204=100 ;2ND SET-UP CLEARANCE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M9
15 L Z+100 FMAX M2
```
3.5 BORING (Cycle 202, DIN/ISO: G202)

Cycle run

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

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

3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.

4. The TNC then orients the spindle to the position that is defined in parameter Q336.

5. If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).

6. The tool then retracts to set-up clearance at the retraction rate, and from there—if programmed—to the 2nd set-up clearance at $F_{\text{MAX}}$. If Q214=0 the tool point remains on the wall of the hole.

7. The TNC finally positions the tool back at the center of the hole.
Please note while programming:

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

Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed. After machining, the TNC positions the tool back at the starting point of the machining plane. This way, you can continue positioning incrementally. If the functions M7 or M8 were active before calling the cycle, the TNC will reconstruct this previous state at the end of the cycle.

- Danger of collision! Use the machine parameter displayDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off). Keep in mind that the TNC reverses the calculation for pre-positioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface! Select a disengaging direction in which the tool moves away from the edge of the hole. Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. During retraction the TNC automatically takes an active rotation of the coordinate system into account.
Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during boring at mm/min. Input range 0 to 99999.9999, 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
- **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
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the TNC retracts the tool on the hole bottom (after spindle orientation)
  0: Do not retract the tool
  1: Retract the tool in minus direction of the principle axis
  2: Retract the tool in minus direction of the minor axis
  3: Retract the tool in plus direction of the principle axis
  4: Retract the tool in plus direction of the minor axis
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before retracting it. Input range -360.000 to 360.000

```
10 L Z+100 R0 FMAX
11 CYCL DEF 202 BORING
Q200=2 ;SET-UP CLEARANCE
Q201=-15 ;DEPTH
Q206=100 ;FEED RATE FOR PLUNGING
Q211=0.5 ;DWELL TIME AT BOTTOM
Q208=250 ;RETRACTION FEED RATE
Q203=+20 ;SURFACE COORDINATE
Q204=100 ;2ND SET-UP CLEARANCE
Q214=1 ;DISENGAGING DIRECTN
Q336=0 ;ANGLE OF SPINDLE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
```
3.6 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)

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

Please note while programming:

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

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

\begin{itemize}
  \item \textbf{Danger of collision!}
  
  Use the machine parameter \textit{displayDepthErr} to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off). Keep in mind that the TNC reverses the calculation for pre-positioning when a \textbf{positive depth is entered}. This means that the tool moves at rapid traverse in the tool axis to set-up clearance \textbf{below} the workpiece surface!
\end{itemize}
Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.9999; 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 set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999
- **No. Breaks before retracting** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip removal. For chip breaking, the TNC retracts the tool each time by the value in Q256. Input range 0 to 99999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999

### NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2</td>
<td>SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q201=-20</td>
<td>DEPTH</td>
</tr>
<tr>
<td>Q206=150</td>
<td>FEED RATE FOR PLUNGING</td>
</tr>
<tr>
<td>Q202=5</td>
<td>PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q211=0</td>
<td>DWELL TIME AT TOP</td>
</tr>
<tr>
<td>Q203=+20</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50</td>
<td>2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q212=0.2</td>
<td>DECREMENT</td>
</tr>
<tr>
<td>Q213=3</td>
<td>CHIP BREAKING</td>
</tr>
<tr>
<td>Q205=3</td>
<td>MIN. PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q211=0.25</td>
<td>DWELL TIME AT BOTTOM</td>
</tr>
<tr>
<td>Q208=500</td>
<td>RETRACTION FEED RATE</td>
</tr>
<tr>
<td>Q256=0.2</td>
<td>DIST. FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q395=0</td>
<td>DEPTH REFERENCE</td>
</tr>
</tbody>
</table>
Fixed Cycles: Drilling

3.6 UNIVERSAL DRILLING (Cycle 203)

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

- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.999, alternatively FMAX, FAUTO

- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999

- **Depth reference** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the tip angle of the tool must be defined in the T ANGLE column of the TOOL.T tool table.
  - 0 = depth in relation to the tool tip
  - 1 = depth in relation to the cylindrical part of the tool
3.7 BACK BORING (Cycle 204, DIN/ISO: G204)

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

1. The TNC positions the tool in the tool axis at rapid traverse $F_{MAX}$ to set-up clearance above the workpiece surface.

2. The TNC then orients the spindle to the 0° position with an oriented spindle stop and displaces the tool by the off-center distance.

3. The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached set-up clearance on the underside of the workpiece.

4. The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.

5. If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. The TNC carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.

6. The tool then retracts to set-up clearance at the feed rate for pre-positioning, and from there—if programmed—to the 2nd set-up clearance at $F_{MAX}$.

7. The TNC finally positions the tool back at the center 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.
- 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 \texttt{R0}.

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

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

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

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

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

Danger of collision!

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

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

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

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

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

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

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

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

- **Dwell time** Q255: Dwell time in seconds at the top of the bore hole. Input range 0 to 3600.0000

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

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

- **Disengaging direction (1/2/3/4)** Q214: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation); programming 0 is not allowed
  1. Retract the tool in minus direction of the principle axis
  2. Retract the tool in minus direction of the minor axis
  3. Retract the tool in plus direction of the principle axis
  4. Retract the tool in plus direction of the minor axis

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

### NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 204 BACK BORING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q249=+5 ;DEPTH OF COUNTERBORE</td>
</tr>
<tr>
<td>Q250=20 ;MATERIAL THICKNESS</td>
</tr>
<tr>
<td>Q251=3.5 ;OFF-CENTER DISTANCE</td>
</tr>
<tr>
<td>Q252=15 ;TOOL EDGE HEIGHT</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q254=200 ;F COUNTERBORING</td>
</tr>
<tr>
<td>Q255=0 ;Dwell TIME</td>
</tr>
<tr>
<td>Q203=+20 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q214=1 ;DISENAGING DIRECTN</td>
</tr>
<tr>
<td>Q336=0 ;ANGLE OF SPINDLE</td>
</tr>
</tbody>
</table>
Fixed Cycles: Drilling

3.8 UNIVERSAL PECKING (Cycle 205)

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

Cycle run

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

2. If you enter a deepened starting point, the TNC move at the defined positioning feed rate to the set-up clearance above the deepened starting point.

3. The tool drills to the first plunging depth at the entered feed rate \(F\).

4. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at \(F_{MAX}\) 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 set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up 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 R0.

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

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

If you use Q379 to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. The TNC does not change retracting movements; they are referenced to the coordinate of the workpiece surface.

Danger of collision!

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

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

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.9999; 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
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202. Input range 0 to 99999.9999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999
- **Upper advanced stop distance** Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth. Input range 0 to 99999.9999
- **Lower advanced stop distance** Q259 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth. Input range 0 to 99999.9999
- **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.9999
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000

NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 205 UNIVERSAL PECKING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q201=-80 ;DEPTH</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q202=15 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q203=+100 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q212=0.5 ;DECREMENT</td>
</tr>
<tr>
<td>Q205=3 ;MIN. PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q258=0.5 ;UPPER ADV. STOP DIST.</td>
</tr>
<tr>
<td>Q259=1 ;LOWER ADV. STOP DIST.</td>
</tr>
<tr>
<td>Q257=5 ;DEPTH FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q256=0.2 ;DIST. FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q211=0.25 ;DWELL TIME AT BOTTOM</td>
</tr>
<tr>
<td>Q379=7.5 ;START POINT</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q208=9999 ;RETRACTION FEED RATE</td>
</tr>
<tr>
<td>Q395=0 ;DEPTH REFERENCE</td>
</tr>
</tbody>
</table>
Deepened starting point Q379 (incremental with respect to the workpiece surface): Starting position for actual drilling operation. The TNC moves at the feed rate for pre-positioning from the set-up clearance above the workpiece surface to the set-up clearance above the deepened starting point. Input range 0 to 99999.9999

Feed rate for pre-positioning Q253: Defines the traversing speed of the tool when returning to the plunging depth after having retracted for chip breaking (Q256). This feed rate is also effective when the tool is positioned to a deepened starting point (Q379 not equal to 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively FMAX, FAUTO

Feed rate for retraction Q208: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.9999, alternatively FMAX, FAUTO

Depth reference Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the tip angle of the tool must be defined in the T ANGLE column of the TOOL T tool table.

0 = depth in relation to the tool tip
1 = depth in relation to the cylindrical part of the tool
3.9 BORE MILLING (Cycle 208)

Cycle run
1. The TNC positions the tool in the tool axis at rapid traverse $F_{MAX}$ to the programmed set-up clearance above the workpiece surface and then moves the tool to the bore hole circumference on a rounded arc (if enough space is available).
2. The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate $F$.
3. When the drilling depth is reached, the TNC once again traverses a full circle to remove the material remaining after the initial plunge.
4. The TNC then positions the tool at the center of the hole again.
5. Finally the TNC returns to the setup clearance at $F_{MAX}$. If programmed, the tool moves to the 2nd set-up 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 \( \text{DEPTH} = 0 \), 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 \( \text{ANGLE} \) column of the tool table. The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

Danger of collision!

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

Keep in mind that the TNC reverses the calculation for pre-positioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!
3.9 BORE MILLING (Cycle 208)

Cycle parameters

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

NC blocks

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

Cycle run

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

2. Then the TNC moves the tool at the defined positioning feed rate to the set-up clearance above the deepened starting point and activates the drilling speed ($M3$) and the coolant. The TNC executes the approach motion with the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.

3. The tool drills to the hole depth at the feed rate $F$, or to the plunging depth if a smaller infeed value has been entered. The plunging depth is decreased after each infeed by the decrement. If you have entered a dwell depth, the TNC reduces the feed rate by the feed rate factor after the dwell depth has been reached.

4. If programmed, the tool remains at the hole bottom for chip breaking.

5. The TNC repeats this process (3 to 4) until the programmed total hole depth is reached.

6. After the TNC has reached the hole depth, the TNC switches off the coolant and resets the drilling speed to the value defined for retraction.

7. The tool is retracted to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at $F_{\text{MAX}}$.

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!

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

3.10 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)

Cycle parameters

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

NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q201=-80 ;DEPTH</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q211=0.25 ;DWELL TIME AT BOTTOM</td>
</tr>
<tr>
<td>Q203=+100;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q208=1000;RETRACTION FEED RATE</td>
</tr>
<tr>
<td>Q426=3 ;DIR. OF SPINDLE ROT.</td>
</tr>
<tr>
<td>Q427=25 ;ROT. SPEED INFEED/OUT</td>
</tr>
<tr>
<td>Q428=500 ;DRILLING SPEED</td>
</tr>
<tr>
<td>Q429=8 ;COOLANT ON</td>
</tr>
<tr>
<td>Q430=9 ;COOLANT OFF</td>
</tr>
<tr>
<td>Q435=0 ;DEWELL DEPTH</td>
</tr>
<tr>
<td>Q401=100 ;FEED RATE FACTOR</td>
</tr>
<tr>
<td>Q202=9999;MAX. PLUNGING DEPTH PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q212=0 ;DECREMENT</td>
</tr>
<tr>
<td>Q205=0 ;MIN. PLUNGING DEPTH PLUNGING DEPTH</td>
</tr>
</tbody>
</table>
M function for coolant on? Q429: M function for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at the deepened starting point. Input range 0 to 999

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

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

Feed rate factor Q401: Factor by which the TNC reduces the feed rate after the dwell depth has been reached. Input range 0 to 100

Plunging depth Q202 (incremental): Infeed per cut. The depth does not have to be a multiple of the plunging depth. Input range 0 to 99999.9999

Decrement Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999

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

**Example: Drilling cycles**

```
0 BEGIN PGM C200 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20  Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0  
3 TOOL CALL 1 Z S4500  Tool call (tool radius 3)
4 L Z+250 R0 FMAX  Retract the tool
5 CYCL DEF 200 DRILLING  Cycle definition
   Q200=2 ;SET-UP CLEARANCE
   Q201=-15 ;DEPTH
   Q206=250 ;FEED RATE FOR PLUNGING
   Q202=5 ;PLUNGING DEPTH
   Q210=0 ;DWELL TIME AT TOP
   Q203=-10 ;SURFACE COORDINATE
   Q204=20 ;2ND SET-UP CLEARANCE
   Q211=0.2 ;DWELL TIME AT BOTTOM
   Q395=0 ;DEPTH REFERENCE
6 L X+10 Y+10 R0 FMAX M3  Approach hole 1, spindle ON
7 CYCL CALL  Cycle call
8 L Y+90 R0 FMAX M99  Approach hole 2, call cycle
9 L X+90 R0 FMAX M99  Approach hole 3, call cycle
10 L Y+10 R0 FMAX M99  Approach hole 4, call cycle
11 L Z+250 R0 FMAX M2  Retract the tool, end program
12 END PGM C200 MM
```
Example: Using drilling cycles in connection with PATTERN DEF

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

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

**Program sequence**
- Centering (tool radius 4)
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)

```plaintext
0 BEGIN PGM 1 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20; Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0
3 TOOL CALL 1 Z S5000; Call the centering tool (tool radius 4)
4 L Z+10 R0 F5000; Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
5 PATTERN DEF; Define all drilling positions in the point pattern
   POS1(X+10 Y+10 Z+0)
   POS2(X+40 Y+30 Z+0)
   POS3(X+20 Y+55 Z+0)
   POS4(X+10 Y+90 Z+0)
   POS5(X+90 Y+90 Z+0)
   POS6(X+80 Y+65 Z+0)
   POS7(X+80 Y+30 Z+0)
   POS8(X+90 Y+10 Z+0)
6 CYCL DEF 240 CENTERING; Cycle definition: CENTERING
   Q200=2; SET-UP CLEARANCE
   Q343=0; SELECT DEPTH/DIA.
   Q201=-2; DEPTH
   Q344=-10; DIAMETER
   Q206=150; FEED RATE FOR PLNGNG
   Q211=0; DWELL TIME AT BOTTOM
   Q203=+0; SURFACE COORDINATE
   Q204=50; 2ND SET-UP CLEARANCE
7 CYCL CALL PAT F5000 M13; Call the cycle in connection with the hole pattern
8 L Z+100 R0 FMAX; Retract the tool, change the tool
9 TOOL CALL 2 Z S5000; Call the drilling tool (radius 2.4)
10 L Z+10 R0 F5000; Move tool to clearance height (enter a value for F)
```
## Fixed Cycles: Drilling

### 3.11 Programming Examples

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

---

**Q200** = 2: Set-up clearance for drilling

**Q201** = -25: Depth of the drill

**Q206** = 150: Feeding rate for plunging

**Q202** = 5: Depth of plunging

**Q211** = 0: Dwell time at the top

**Q203** = +0: Surface coordinate

**Q204** = 50: Second set-up clearance

**Q211** = 0.2: Dwell time at the bottom

**Q395** = 0: Depth reference
Fixed Cycles:
Tapping / Thread Milling
# 4 Fixed Cycles: Tapping / Thread Milling

## 4.1 Fundamentals

### Overview

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

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>206</td>
<td>206 TAPPING NEW</td>
<td>107</td>
</tr>
<tr>
<td></td>
<td>With a floating tap holder, with automatic prepositioning, 2nd Set-up clearance</td>
<td></td>
</tr>
<tr>
<td>207</td>
<td>207 TAPPING NEW</td>
<td>110</td>
</tr>
<tr>
<td></td>
<td>Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance</td>
<td></td>
</tr>
<tr>
<td>209</td>
<td>209 TAPPING WITH CHIP BREAKING</td>
<td>113</td>
</tr>
<tr>
<td></td>
<td>Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking</td>
<td></td>
</tr>
<tr>
<td>262</td>
<td>262 THREAD MILLING</td>
<td>119</td>
</tr>
<tr>
<td></td>
<td>Cycle for milling a thread in pre-drilled material</td>
<td></td>
</tr>
<tr>
<td>263</td>
<td>263 THREAD MILLING/ COUNTERSINKING</td>
<td>122</td>
</tr>
<tr>
<td></td>
<td>Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer</td>
<td></td>
</tr>
<tr>
<td>264</td>
<td>264 THREAD DRILLING/MILLING</td>
<td>126</td>
</tr>
<tr>
<td></td>
<td>Cycle for drilling into solid material with subsequent milling of the thread with a tool</td>
<td></td>
</tr>
<tr>
<td>265</td>
<td>265 HELICAL THREAD DRILLING/ MILLING</td>
<td>130</td>
</tr>
<tr>
<td></td>
<td>Cycle for milling the thread into solid material</td>
<td></td>
</tr>
<tr>
<td>267</td>
<td>267 OUTSIDE THREAD MILLING</td>
<td>134</td>
</tr>
<tr>
<td></td>
<td>Cycle for milling an external thread and machining a countersunk chamfer</td>
<td></td>
</tr>
</tbody>
</table>
4.2 TAPPING with a floating tap holder (Cycle 206, DIN/ISO: G206)

Cycle run

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

2. The tool drills to the total hole depth in one movement.

3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at $F_{\text{MAX}}$.

4. At the set-up clearance, the direction of spindle rotation reverses once again.
4.2 TAPPING with a floating tap holder (Cycle 206)

Please note while programming:

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.
- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.
- When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).
- For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.
- If you enter the thread pitch of the tap in the Pitch column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match. In Cycle 206 the TNC uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

Danger of collision!

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

- **Set-up clearance Q200** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
  
  Guide value: 4x pitch.

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

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

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

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

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.

---

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 CYCL DEF 206 TAPPING NEW</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q201=-20 ;THREAD DEPTH</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q211=0.25 ;DWELL TIME AT BOTTOM</td>
</tr>
<tr>
<td>Q203=+25 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
</tbody>
</table>
Fixed Cycles: Tapping / Thread Milling

4.3 RIGID TAPPING without a floating tap holder (Cycle 207)

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

Cycle run

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

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

2. The tool drills to the total hole depth in one movement.

3. It then reverses the direction of spindle rotation again and the tool is retracted to the setup clearance. If you have entered a 2nd set-up clearance the TNC will move the tool with $F_{\text{MAX}}$ towards it.

4. The TNC stops the spindle turning at set-up clearance.
Please note while programming:

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

Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.
- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The TNC calculates the feed rate from the spindle speed. If the feed-rate override is used during tapping, the TNC automatically adjusts the feed rate.
- The feed-rate override knob is disabled.
- At the end of the cycle the spindle comes to a stop.
- Before the next operation, restart the spindle with M3 (or M4).
- If you enter the thread pitch of the tap in the Pitch column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match.

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

4.3 RIGID TAPPING without a floating tap holder (Cycle 207)

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - - = left-hand thread
  - Input range -99.9999 to 99.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

Retracting after a program interruption

Retracting in the Manual Operation mode

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

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

You can interrupt the thread cutting process by pressing the NC Stop key. The TNC displays the MANUAL TRAVERSE soft key. After you pressed the MANUAL TRAVERSE soft key, you can retract the tool from the active spindle axis. To resume machining after the interruption, press the RESTORE POSITION soft key and NC Start. The TNC moves the tool back to the position it had assumed before the NC Stop key was pressed.

When retracting the tool you can move it in the positive and negative tool axis directions. Please keep this in mind during retraction—danger of collision!
4.4 TAPPING WITH CHIP BREAKING
(Cycle 209, DIN/ISO: G209)

Cycle run

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

1 The TNC positions the tool in the tool axis at rapid traverse $F_{MAX}$ to the programmed set-up clearance above the workpiece surface. There it carries out an oriented spindle stop.

2 The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition. If you have defined a factor for increasing the spindle speed, the TNC retracts from the hole at the corresponding speed.

3 It then reverses the direction of spindle rotation again and advances to the next infeed depth.

4 The TNC repeats this process (2 to 3) until the programmed thread depth is reached.

5 The tool is then retracted to set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at $F_{MAX}$.

6 The TNC stops the spindle turning at set-up clearance.
Please note while programming:

- Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.
- This cycle is effective only for machines with servo-controlled spindle.
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation \( R_0 \).
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.
- The TNC calculates the feed rate from the spindle speed. If the feed-rate override is used during tapping, the TNC automatically adjusts the feed rate.
- The feed-rate override knob is disabled.
- If you defined an rpm factor for fast retraction in cycle parameter \( Q403 \), the TNC limits the speed to the maximum speed of the active gear range.
- At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with \( M3 \) (or \( M4 \)).
- If you enter the thread pitch of the tap in the Pitch column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match.

**Danger of collision!**

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

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - - = left-hand thread
  - Input range -99.9999 to 99.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256: The TNC multiplies the pitch Q239 by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter Q256 = 0, the TNC retracts the tool completely from the hole (to the set-up clearance) for chip breaking. Input range 0.000 to 99999.9999
- **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. Maximum increase to maximum speed of the active gear range.

NC blocks

<table>
<thead>
<tr>
<th>26 CYCL DEF 209 TAPPING W/ CHIP BRKG</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q201=-20 ;DEPTH</td>
</tr>
<tr>
<td>Q239=+1 ;THREAD PITCH</td>
</tr>
<tr>
<td>Q204=+25 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q203=+25 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q257=5 ;DEPTH FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q256=+1 ;DIST. FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q336=50 ;ANGLE OF SPINDLE</td>
</tr>
<tr>
<td>Q403=1.5 ;RPM FACTOR</td>
</tr>
</tbody>
</table>
Fixed Cycles: Tapping / Thread Milling
4.4 TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209)

Retracting after a program interruption

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

Retracting in the Program Run, Single Block or Full Sequence mode
You can interrupt the thread cutting process by pressing the NC Stop key. The TNC displays the MANUAL TRAVERSE soft key. After you pressed the MANUAL TRAVERSE soft key, you can retract the tool from the active spindle axis. To resume machining after the interruption, press the RESTORE POSITION soft key and NC Start. The TNC moves the tool back to the position it had assumed before the NC Stop key was pressed.

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

Prerequisites

- Your machine tool should feature internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars).
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius \( DR \) in the TOOL CALL.
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign \( Q239 (+ = \text{right-hand thread} / – = \text{left-hand thread}) \) and milling method \( Q351 (+1 = \text{climb} / –1 = \text{up-cut}) \). 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 MIRROR IMAGE in only one axis.
Danger of collision!
Always program the same algebraic sign for the infeeds: Cycles comprise several sequences of operation that are independent of each other. The order of precedence according to which the work direction is determined is described with the individual cycles. For example, if you only want to repeat the countersinking process of a cycle, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

Procedure in case of a tool break
If a tool break occurs during thread cutting, stop program run, change to the Positioning with MDI operating mode and move the tool on a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.
4.6 THREAD MILLING (Cycle 262, DIN/ISO: G262)

Cycle run

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

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

3. The tool then moves tangentially on a helical path to the thread major diameter. Before the helical approach, a compensating motion of the tool axis is carried out in order to begin at the programmed starting plane for the thread path.

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

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

6. At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.
Please note while programming:

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

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

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

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

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

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

Danger of collision!

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

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

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - - = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Threads per step** Q355: Number of thread starts by which the tool is displaced:
  - 0 = one helix on the thread depth
  - 1 = continuous helix on the complete thread length
  - >1 = several helix paths with approach and departure, between these the TNC sets the tool by Q355 x pitch. Input range 0 to 99999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively FMAX, FAUTO
- **Climb or up-cut** Q351: Type of milling operation with M3
  - +1 = Climb
  - -1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.9999 alternatively FAUTO
- **Feed rate for approaching** Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.9999 alternatively FAUTO

### NC blocks

<table>
<thead>
<tr>
<th>CYCL DEF 262 THREAD MILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q335=10 ; NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q239=+1.5 ; THREAD PITCH</td>
</tr>
<tr>
<td>Q201=-20 ; THREAD DEPTH</td>
</tr>
<tr>
<td>Q355=0 ; THREADS PER STEP</td>
</tr>
<tr>
<td>Q253=750 ; F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q351=+1 ; CLIMB OR UP-CUT</td>
</tr>
<tr>
<td>Q200=2 ; SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+30 ; SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ; 2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q207=500 ; FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q512=0 ; FEED RATE FOR APPROACHING</td>
</tr>
</tbody>
</table>
4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO:G263)

Cycle run
1. The TNC positions the tool in the tool axis at rapid traverse \( F_{\text{MAX}} \) to the entered set-up clearance above the workpiece surface.

Countersinking
2. The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up clearance, and then at the feed rate for countersinking to the countersinking depth.
3. If a safety clearance to the side has been entered, the TNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth.
4. Then, depending on the available space, the TNC makes a tangential approach to the core diameter, either tangentially from the center or with a pre-positioning move to the side, and follows a circular path.

Countersinking at front
5. The tool moves at the feed rate for pre-positioning to the sinking depth at front.
6. The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
7. The tool then moves in a semicircle to the hole center.

Thread milling
8. The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).
9. Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
10. After that the tool departs the contour tangentially and returns to the starting point in the working plane.
11. At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.
Please note while programming:

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

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

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

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

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

Danger of collision!

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

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


Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Countersinking depth** Q356 (incremental): Distance between tool tip and the top surface of the workpiece. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively FMAX, FAUTO
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  –1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Set-up clearance to the side** Q357 (incremental): Distance between tool tooth and the wall of the hole. Input range 0 to 99999.9999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999.
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999.
- **Feed rate for countersinking** Q254: Traversing speed of the tool during countersinking in mm/min. Input range 0 to 99999.9999 alternatively FAUTO, FU.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.9999 alternatively FAUTO.
- **Feed rate for approaching** Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.9999 alternatively FAUTO.

### NC blocks

<table>
<thead>
<tr>
<th>25 CYCL DEF 263 THREAD MLLNG/ CNTSNKG</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q335=10 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q239=+1.5 ;THREAD PITCH</td>
</tr>
<tr>
<td>Q201=-16 ;THREAD DEPTH</td>
</tr>
<tr>
<td>Q356=-20 ;COUNTERSINKING DEPTH</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q351=+1 ;CLimb OR UP-CUT</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q357=0.2 ;CLEARANCE TO SIDE</td>
</tr>
<tr>
<td>Q358=+0 ;DEPTH AT FRONT</td>
</tr>
<tr>
<td>Q359=+0 ;OFFSET AT FRONT</td>
</tr>
<tr>
<td>Q203=+30 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q254=150 ;F COUNTERBORING</td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q512=0 ;FEED RATE FOR APPROACHING</td>
</tr>
</tbody>
</table>
4.8 THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264)

**Cycle run**
1. The TNC positions the tool in the tool axis at rapid traverse \( F_{MAX} \) to the entered set-up clearance above the workpiece surface.

**Drilling**
2. The tool drills to the first plunging depth at the programmed feed rate for plunging.
3. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at \( F_{MAX} \) 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 sinking depth at front.
7. The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
8. The tool then moves in a semicircle to the hole center.

**Thread milling**
9. The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).
10. Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
11. After that the tool departs the contour tangentially and returns to the starting point in the working plane.
12. At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.
Please note while programming:

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

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

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

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

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

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 when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively `FMAX`, `FAUTO`
- **Climb or up-cut** `Q351`: Type of milling operation with M3
  - `+1` = Climb
  - `-1` = Up-cut (If you enter 0, climb milling is used for machining)
- **Plunging depth** `Q202` (incremental): Infeed per cut. The depth does not have to be a multiple of the plunging depth. Input range 0 to 99999.9999

The TNC will go to depth in one movement if:
- the plunging depth is equal to the depth
- the plunging depth is greater than the depth
Upper advanced stop distance Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole. Input range 0 to 99999.9999

Infeed depth for chip breaking Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999

Retraction rate for chip breaking Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.9999

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

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

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

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

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

Feed rate for plunging Q206: Traversing speed of the tool when moving into the workpiece in mm/min. Input range 0 to 99999.9999 alternatively FAUTO, FU

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

Feed rate for approaching Q206: Traversing speed of the tool in mm/min while approaching. If the thread diameters are small, you can reduce the danger of tool breakage by using a reduced approaching feed rate. Input range 0 to 99999.9999 alternatively FAUTO

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 CYCL DEF 264 THREAD DRILLNG/MLLNG</td>
</tr>
<tr>
<td>Q335=10 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q239=+1.5 ;THREAD PITCH</td>
</tr>
<tr>
<td>Q201=-16 ;THREAD DEPTH</td>
</tr>
<tr>
<td>Q356=-20 ;TOTAL HOLE DEPTH</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q351=+1 ;CLIMB OR UP-CUT</td>
</tr>
<tr>
<td>Q202=5 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q258=0.2 ;ADV. STOP DIST.</td>
</tr>
<tr>
<td>Q257=5 ;DEPTH FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q256=0.2 ;DIST. FOR CHIP BRKNG</td>
</tr>
<tr>
<td>Q358=0 ;DEPTH AT FRONT</td>
</tr>
<tr>
<td>Q359=0 ;OFFSET AT FRONT</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+30 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q512=0 ;FEED RATE FOR APPROACHING</td>
</tr>
</tbody>
</table>
4.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265)

Cycle run
1. The TNC positions the tool in the tool axis at rapid traverse $F_{MAX}$ to the entered set-up clearance above the workpiece surface.

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

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

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

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:

1. Thread depth
2. Depth at front

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

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

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

Danger of collision!

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

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

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

Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  - + = right-hand thread
  - - = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively **FMAX, FAUTO**
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- **Countersinking** Q360: Running the chamfer
  - 0 = before thread milling
  - 1 = after thread milling
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265) 4.9

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

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

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

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 CYCL DEF 265 HEL. THREAD DRLG/MLG</td>
</tr>
<tr>
<td>Q335=10 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q239=+1.5 ;THREAD PITCH</td>
</tr>
<tr>
<td>Q201=-16 ;THREAD DEPTH</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q358=+0 ;DEPTH AT FRONT</td>
</tr>
<tr>
<td>Q359=+0 ;OFFSET AT FRONT</td>
</tr>
<tr>
<td>Q360=0 ;COUNTERSINKING</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+30 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q254=150 ;F COUNTERBORING</td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
</tr>
</tbody>
</table>
4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267)

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

Countersinking at front
2. The TNC moves on the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.
3. The tool moves at the feed rate for pre-positioning to the sinking depth at front.
4. The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
5. The tool then moves in a semicircle to the starting point.

Thread milling
6. The TNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.
7. The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
8. The tool then approaches the thread diameter tangentially in a helical movement.
9. Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
10. After that the tool departs the contour tangentially and returns to the starting point in the working plane.
11. At the end of the cycle, the TNC retracts the tool in rapid traverse to setup clearance or, if programmed, to the 2nd setup clearance.
Please note while programming:

- Program a positioning block for the starting point (stud center) in the working plane with radius compensation R0.
- The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).
- The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Thread depth
  2. Depth at front
- If you program a depth parameter to be 0, the TNC does not execute that step.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.

**Danger of collision!**

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

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

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. Input range -99999.9999 to 99999.9999
- **Threads per step** Q355: Number of thread starts by which the tool is displaced:
  0 = one helix on the thread depth
  1 = continuous helix on the complete thread length
  >1 = several helix paths with approach and departure, between these the TNC sets the tool by Q355 x pitch. Input range 0 to 99999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min. Input range 0 to 99999.9999 alternatively FMAX, FAUTO
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (If you enter 0, climb milling is used for machining)
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267) 4.10

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

NC blocks

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

Example: Thread milling

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

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

Program sequence
- Centering
- Drilling
- Tapping

Program sequence

0 BEGIN PGM 1 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20
2 BLK FORM 0.2 X+100 Y+100 Y+0
3 TOOL CALL 1 Z S5000
4 L Z+10 R0 F5000
5 SEL PATTERN "TAB1"
6 CYCL DEF 240 CENTERING
    Q200=2 ;SET-UP CLEARANCE
    Q343=1 ;SELECT DIA./DEPTH
    Q201=-3.5 ;DEPTH
    Q344=-7 ;DIAMETER
    Q206=150 ;FEED RATE FOR PLNGNG
    Q11=0 ;DWELL TIME AT DEPTH
    Q203=+0 ;SURFACE COORDINATE
    Q204=0 ;2ND SET-UP CLEARANCE
10 CYCL CALL PAT F5000 M3
11 L Z+100 R0 FMAX M6
12 TOOL CALL 2 Z S5000
13 L Z+10 R0 F5000
14 CYCL DEF 200 DRILLING
    Q200=2 ;SET-UP CLEARANCE
    Q201=-25 ;DEPTH
    Q206=150 ;FEED RATE FOR PLNGNG
    Q202=5 ;PLUNGING DEPTH
    Q210=0 ;DWELL TIME AT TOP
    Q203=+0 ;SURFACE COORDINATE
| Q204=0 | ;2ND SET-UP CLEARANCE | 0 must be entered here, effective as defined in point table |
| Q211=0.2 | ;DWELL TIME AT DEPTH |
| Q395=0 | ;DEPTH REFERENCE |
| 15 CYCL CALL PAT F5000 M3 | Cycle call in connection with point table TAB1.PNT |
| 16 L Z+100 R0 FMAX M6 | Retract the tool, change the tool |
| 17 TOOL CALL 3 Z 5200 | Call tool: tap |
| 18 L Z+50 R0 FMAX | Move tool to clearance height |
| 19 CYCL DEF 206 TAPPING | Cycle definition for tapping |
| Q200=2 | ;SET-UP CLEARANCE |
| Q201=-25 | ;DEPTH OF THREAD |
| Q206=150 | ;FEED RATE FOR PLNGNG |
| Q211=0 | ;DWELL TIME AT DEPTH |
| Q203=+0 | ;SURFACE COORDINATE | 0 must be entered here, effective as defined in point table |
| Q204=0 | ;2ND SET-UP CLEARANCE | 0 must be entered here, effective as defined in point table |
| 20 CYCL CALL PAT F5000 M3 | Cycle call in connection with point table TAB1.PNT |
| 21 L Z+100 R0 FMAX M2 | Retract the tool, end program |
| 22 END PGM 1 MM |

**Point table TAB1.PNT**

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

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

### Overview

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

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>251</td>
<td>251 RECTANGULAR POCKET</td>
<td>143</td>
</tr>
<tr>
<td></td>
<td>Roughing/finishing cycle with selection of machining operation and helical plunging</td>
<td></td>
</tr>
<tr>
<td>252</td>
<td>252 CIRCULAR POCKET</td>
<td>147</td>
</tr>
<tr>
<td></td>
<td>Roughing/finishing cycle with selection of machining operation and helical plunging</td>
<td></td>
</tr>
<tr>
<td>253</td>
<td>253 SLOT MILLING</td>
<td>152</td>
</tr>
<tr>
<td></td>
<td>Roughing/finishing cycle with selection of machining operation and reciprocal plunging</td>
<td></td>
</tr>
<tr>
<td>254</td>
<td>254 CIRCULAR SLOT</td>
<td>156</td>
</tr>
<tr>
<td></td>
<td>Roughing/finishing cycle with selection of machining operation and reciprocal plunging</td>
<td></td>
</tr>
<tr>
<td>256</td>
<td>256 RECTANGULAR STUD</td>
<td>161</td>
</tr>
<tr>
<td></td>
<td>Roughing/finishing cycle with stepover, if multiple passes are required</td>
<td></td>
</tr>
<tr>
<td>257</td>
<td>257 CIRCULAR STUD</td>
<td>165</td>
</tr>
<tr>
<td></td>
<td>Roughing/finishing cycle with stepover, if multiple passes are required</td>
<td></td>
</tr>
<tr>
<td>233</td>
<td>233 FACE MILLING</td>
<td>174</td>
</tr>
<tr>
<td></td>
<td>Machining the face with up to 3 limits</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 the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter Q366.
2. The TNC roughs out the pocket from the inside out, taking the overlap factor (parameter Q370) and the finishing allowance (parameters Q368 and Q369) into account.
3. At the end of the roughing operation, the TNC moves the tool tangentially away from the pocket wall, then moves by the set-up clearance above the current plunging depth and returns from there at rapid traverse to the pocket center.
4. This process is repeated until the programmed pocket depth is reached.

**Finishing**

5. If finishing allowances are defined, the tool plunges the workpiece at the pocket center and moves to the plunging depth for finishing. The TNC first finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.
6. Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.
5.2 RECTANGULAR POCKET (Cycle 251)

Please note while programming:

- With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.
- Pre-position the tool in the machining plane to the starting position with radius compensation R0. Note parameter Q367 (position).
- The TNC automatically pre-positions the tool in the tool axis. Note the 2nd set-up clearance Q204.
- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- At the end of the cycle, the TNC returns the tool to the starting position.
- At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.
- The TNC outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the suppressPlungeErr machine parameter.
- The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

Danger of collision!

- Use the machine parameter displayDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).
- Keep in mind that the TNC reverses the calculation for pre-positioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!
- If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.
**Cycle parameters**

- **Machining operation (0/1/2) Q215**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing

  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined.

- **1st side length Q218 (incremental)**: Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999

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

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

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

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

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

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

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

  **PREDEF**: The TNC uses the value from the GLOBAL DEF block (if you enter 0, climb milling is used for machining)

- **Depth Q201 (incremental)**: Distance between workpiece surface and bottom of 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
5.2 RECTANGULAR POCKET (Cycle 251)

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

### NC blocks

<table>
<thead>
<tr>
<th>NC Blocks</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 CYCL DEF 251 RECTANGULAR POCKET</td>
<td></td>
</tr>
<tr>
<td>Q215=0 ;MACHINING OPERATION</td>
<td></td>
</tr>
<tr>
<td>Q218=80 ;FIRST SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td>Q219=60 ;2ND SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td>Q220=5 ;CORNER RADIUS</td>
<td></td>
</tr>
<tr>
<td>Q368=0.2 ;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td>Q224=+0 ;ANGLE OF ROTATION</td>
<td></td>
</tr>
<tr>
<td>Q367=0 ;POCKET POSITION</td>
<td></td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>Q351=+1 ;CLIMB OR UP-CUT</td>
<td></td>
</tr>
<tr>
<td>Q201=20 ;DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q202=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q369=0.1 ;ALLOWANCE FOR FLOOR</td>
<td></td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLUNGING</td>
<td></td>
</tr>
<tr>
<td>Q338=5 ;INFEE-- FOR FINISHING</td>
<td></td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q203=+0 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q370=1 ;TOOL PATH OVERLAP</td>
<td></td>
</tr>
<tr>
<td>Q366=1 ;PLUNGE</td>
<td></td>
</tr>
<tr>
<td>Q385=500 ;FINISHING FEED RATE</td>
<td></td>
</tr>
</tbody>
</table>

9 L X+50 Y+50 R0 FMAX M3 M99
5.3  CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)

Cycle run

Use Cycle 252 CIRCULAR POCKET to machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:
- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

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

1. Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified.

2. The TNC positions the tool in the tool axis in front of the pocket wall, taking the finishing allowance Q368 and the set-up clearance Q200 into account.

3. The TNC clears the pocket from the inside out until the diameter Q223 is reached.

4. Then the TNC again positions the tool in the tool axis in front of the pocket wall, taking the finishing allowance Q368 and the set-up clearance Q200 into account, and repeats the finishing process of the pocket wall at the next depth.

5. The TNC repeats this process until the programmed diameter is reached.

6. After machining to the diameter Q223, the TNC retracts the tool tangentially by the finishing allowance Q368 plus the set-up clearance Q200 in the working plane, then retracts at rapid traverse to the set-up clearance Q200 in the tool axis and returns to the pocket center.

7. Next, the TNC moves the tool in the tool axis to the depth Q201 and finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

8. The TNC repeats this process until the depth Q201 plus Q369 is reached.

9. Finally, the tool moves away from the pocket wall tangentially by the set-up clearance Q200, then retracts at rapid traverse to the set-up clearance Q200 in the tool axis and returns at rapid traverse to the pocket center.
Please note while programming:

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

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

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

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

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

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

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

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

Danger of collision!

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

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

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.
Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

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

Cycle parameters

- **Machining operation (0/1/2)** Q215: Define machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined

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

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

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

- **Climb or up-cut** Q351: Type of milling operation with M3
  - +1 = climb
  - -1 = up-cut
  **PREDEF**: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)

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

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

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

- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.9999; alternatively FAUTO, FU, FZ
Infeed for finishing Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999

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

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

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

Path overlap factor Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.9999; alternatively PREDEF

Plunging strategy Q366: Type of plunging strategy:
- 0 = vertical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined as 0 or 90. The TNC will otherwise display an error message.
- 1 = helical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
- Alternative: PREDEF

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

Feed rate reference (0…3) Q439: Define a reference for the programmed feed rate:
0: The feed rate refers to the center point path of the tool
1: The feed rate refers to the tool cutting edge only during side finishing; otherwise, it refers to the center point path
2: The feed rate refers to the tool cutting edge during side and floor finishing; otherwise, it refers to the center point path
3: The feed rate always refers to the tool cutting edge

NC blocks

| NC Blocks | Description
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>8 CYCL DEF 252 CIRCULAR POCKET</td>
<td>MACHINING OPERATION</td>
</tr>
<tr>
<td>Q215=0 ;</td>
<td>CIRCLE DIAMETER</td>
</tr>
<tr>
<td>Q233=60 ;</td>
<td>ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q368=0.2 ;</td>
<td>FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q207=500 ;</td>
<td>CLIMB OR UP-CUT</td>
</tr>
<tr>
<td>Q351=+1 ;</td>
<td>DEPTH</td>
</tr>
<tr>
<td>Q201=−20 ;</td>
<td>PREDEF</td>
</tr>
<tr>
<td>Q202=5 ;</td>
<td>PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q369=0.1 ;</td>
<td>ALLOWANCE FOR FLOOR</td>
</tr>
<tr>
<td>Q206=150 ;</td>
<td>FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q338=5 ;</td>
<td>INFEED FOR FINISHING</td>
</tr>
<tr>
<td>Q200=2 ;</td>
<td>SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+0 ;</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;</td>
<td>2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q370=1 ;</td>
<td>TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q366=1 ;</td>
<td>PLUNGE</td>
</tr>
<tr>
<td>Q385=500 ;</td>
<td>FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q439=3 ;</td>
<td>FEED RATE REFERENCE</td>
</tr>
</tbody>
</table>

L X+50 Y+50 R0 FMAX M3 M99
5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253)

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 and finishing (floor and side finishing)
- Only roughing
- Only finishing (floor 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 programmed 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 left slot arc.
5. Then the TNC finishes the floor of the slot from the inside out.
Please note while programming:

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

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

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

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

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

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

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

Danger of collision!

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

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

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool to the first plunging depth at rapid traverse!
5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253)

Cycle parameters

- Machining operation (0/1/2) Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined

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

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

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

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

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

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

- Climb or up-cut Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (if you enter 0, climb milling is used for machining)

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

- Plunging depth Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
Finishing allowance for floor Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999.

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

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

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

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

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

Plunging strategy Q366: Type of plunging strategy:
- 0 = vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
- 1, 2 = reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
- Alternative: PREDEF.

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

Feed rate reference (0…3) Q439: Define a reference for the programmed feed rate:
- 0: The feed rate refers to the center point path of the tool.
- 1: The feed rate refers to the tool cutting edge only during side finishing; otherwise, it refers to the center point path.
- 2: The feed rate refers to the tool cutting edge during side and floor finishing; otherwise, it refers to the center point path.
- 3: The feed rate always refers to the tool cutting edge.

### NC blocks

<table>
<thead>
<tr>
<th>8 CYCL DEF 253 SLOT MILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q218=80 ;SLOT LENGTH</td>
</tr>
<tr>
<td>Q219=12 ;SLOT WIDTH</td>
</tr>
<tr>
<td>Q368=0.2 ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q374=0.0 ;ANGLE OF ROTATION</td>
</tr>
<tr>
<td>Q367=0 ;SLOT 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=0.1 ;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 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=0 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q366=1 ;PLUNGE</td>
</tr>
<tr>
<td>Q385=500 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q439=0 ;FEED RATE REFERENCE</td>
</tr>
</tbody>
</table>

9 L X+50 Y+50 R0 FMAX M3 M99
5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)

**Cycle run**
Use Cycle 254 to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:
- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

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

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

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

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

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

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

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

Danger of collision!

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

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

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool to the first plunging depth at rapid traverse!
5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)

**Cycle parameters**

- **Machining operation (0/1/2) Q215**: Define machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined

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

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

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

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

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

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

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

- **Angular length Q248** (incremental): Enter the angular length of the slot. Input range 0 to 360.000

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

- **Number of repetitions Q377**: Number of machining operations on a pitch circle. Input range 1 to 99999
Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.9999 alternatively FAUTO, FU, FZ

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

PREDEF: The TNC uses the value from the GLOBAL DEF block (If you enter 0, climb milling is used for machining)

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

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

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

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

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

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

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

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

---

NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=0</td>
<td>MACHINING OPERATION</td>
</tr>
<tr>
<td>Q219=12</td>
<td>SLOT WIDTH</td>
</tr>
<tr>
<td>Q368=0</td>
<td>ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q375=80</td>
<td>PITCH CIRCLE DIA.</td>
</tr>
<tr>
<td>Q367=0</td>
<td>REF. SLOT POSITION</td>
</tr>
<tr>
<td>Q216=-50</td>
<td>CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q217=-50</td>
<td>CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q376=45</td>
<td>STARTING ANGLE</td>
</tr>
<tr>
<td>Q248=90</td>
<td>ANGULAR LENGTH</td>
</tr>
<tr>
<td>Q378=0</td>
<td>STEPPING ANGLE</td>
</tr>
<tr>
<td>Q377=1</td>
<td>NR OF REPETITIONS</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>Q369=0.1</td>
<td>ALLOWANCE FOR FLOOR</td>
</tr>
</tbody>
</table>
### 5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)

- **Plunging strategy** Q366: Type of plunging strategy:
  - 0: vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
  - 1, 2: reciprocal plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message.
  - **PREDEF**: The TNC uses the value from the GLOBAL DEF block.

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

- **Feed rate reference (0…3)** Q439: Define a reference for the programmed feed rate:
  - 0: The feed rate refers to the center point path of the tool.
  - 1: The feed rate refers to the tool cutting edge only during side finishing; otherwise, it refers to the center point path.
  - 2: The feed rate refers to the tool cutting edge during side and floor finishing; otherwise, it refers to the center point path.
  - 3: The feed rate always refers to the tool cutting edge.

---

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q206</td>
<td>150</td>
</tr>
<tr>
<td>Q338</td>
<td>5</td>
</tr>
<tr>
<td>Q385</td>
<td>500</td>
</tr>
<tr>
<td>Q439</td>
<td>0</td>
</tr>
</tbody>
</table>

9 L X+50 Y+50 R0 FMAX M3 M99
5.6 RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)

Cycle run

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

1. The tool moves from the cycle starting position (stud center) to the starting position for stud machining. Specify the starting position with parameter Q437. The standard setting (Q437=0) lies 2 mm to the right next to the stud blank.

2. If the tool is at the 2nd set-up clearance, it moves at rapid traverse FMAX to the set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

3. The tool then moves tangentially to the stud contour and machines one revolution.

4. If the finished dimension cannot be machined with one revolution, the TNC performs a stepover with the current factor, and machines another revolution. The TNC takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached. However, if you chose your starting point at a corner instead of laterally (Q437 not equal 0), the TNC will spirally machine inwards from the starting point until the finished dimension is achieved.

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

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

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

8. At the end of the cycle, the TNC merely positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position.
Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.6 RECTANGULAR STUD (Cycle 256)

Please note while programming:

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

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

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

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

Danger of collision!

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

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

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

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

- **1st side length** Q218: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999

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

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

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

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

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

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

- **Stud position** Q367: Position of the stud in reference to the position of the tool when the cycle is called:
  0: Tool position = stud center
  1: Tool position = left corner below
  2: Tool position = right corner below
  3: Tool position = right corner top
  4: Tool position = left top corner top

- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.9999 alternatively FAUTO, FU, FZ
Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.6 RECTANGULAR STUD (Cycle 256)

- Climb or up-cut Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
- PREDEF: The TNC uses the value from the GLOBAL DEF block (if you enter 0, climb milling is used for machining)
- Depth Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range -9999.9999 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
- Set-up clearance Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively PREDEF
- Coordinate of workpiece surface Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- 2nd set-up clearance Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively PREDEF
- Path overlap factor Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.9999; alternatively PREDEF
- Approach position (0...4) Q437: Define the approach strategy of the tool:
  0: Right of the stud (default setting)
  1: left corner below
  2: right corner below
  3: right corner top
  4: left corner top. If approach marks on the stud surface are caused by the setting Q437=0, specify another approach position.

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 CYCL DEF 256 RECTANGULAR STUD</td>
</tr>
<tr>
<td>Q218=60 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q424=74 ;WORKPC. BLANK SIDE 1</td>
</tr>
<tr>
<td>Q219=40 ;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q425=60 ;WORKPC. BLANK SIDE 2</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 ;STUD 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>Q206=150 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+0 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q370=1 ;TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q437=0 ;APPROACH POSITION</td>
</tr>
<tr>
<td>9 L X+50 Y+50 R0 FMAX M3 M99</td>
</tr>
</tbody>
</table>
5.7 CIRCULAR STUD (Cycle 257, DIN/ISO: G257)

Cycle run

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

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

- Pre-position the tool in the machining plane to the starting position (stud center) with radius compensation R0.
- The TNC automatically pre-positions the tool in the tool axis. Note the 2nd set-up clearance Q204.
- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- At the end of the cycle, the TNC returns the tool to the starting position.
- The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

Danger of collision!

- Use the machine parameter displayDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).
- Keep in mind that the TNC reverses the calculation for pre-positioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!
- The TNC performs an approach motion in this cycle! Depending on the starting angle Q376, the following amount of space must be left next to the stud: At least tool diameter + 2 mm. Danger of collision!
- At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position!
- Enter a starting angle between 0° and 360° in parameter Q376 in order to determine the exact starting position. If you use the default value -1 the TNC automatically calculates the most favorable starting position. These may vary.
Cycle parameters

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

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

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

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

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

- **Starting angle** Q376: Polar angle relative to the stud center from which the tool approaches the stud. Input range 0 to 359°

---

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 CYCL DEF 257 CIRCULAR STUD</td>
<td></td>
</tr>
<tr>
<td>Q223=60</td>
<td>;FINISHED PART DIAMETER</td>
</tr>
<tr>
<td>Q222=60</td>
<td>;WORKPIECE BLANK DIAMETER</td>
</tr>
<tr>
<td>Q368=0.2</td>
<td>;ALLOWANCE FOR SIDE</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>;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q370=1</td>
<td>;TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q376=0</td>
<td>;STARTING ANGLE</td>
</tr>
<tr>
<td>9 L X+50 Y+50 R0 FMAX M3 M99</td>
<td></td>
</tr>
</tbody>
</table>
Cycle run

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

1. If, at the beginning of machining, the work piece is positioned below the 2nd set-up clearance, the TNC will retract the tool back to the 2nd setup clearance.

2. Starting from the center of the stud the TNC moves the tool to the starting point of the stud machining. The starting point depends, among others, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter Q224.

3. The tool moves at rapid traverse $F_{MAX}$ to the setup clearance Q200 and from there with the feed rate for plunging to the first plunging depth.

4. Then the TNC creates the polygon stud in a spiral-shaped pass, taking into account the path overlap.

5. The TNC moves the tool on a tangential path from the outside to the inside.

6. The tool will be lifted in the direction of the spindle axis to the 2nd setup clearance in one rapid movement.

7. If several plunging depths are required, the TNC will position the tool back to the starting point of the stud milling process.

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

9. At the end of the cycle first a departing motion is performed. Then the TNC will move the tool on the tool axis to the 2nd setup clearance.
Please note while programming:

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

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

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

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

Danger of collision!

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

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

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

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

- **Reference circuit** Q573: Definition of whether the dimensioning shall reference to the inscribed circle or to the perimeter:
  - 0 = dimensioning refers to the inscribed circle
  - 1 = dimensioning refers to the perimeter

- **Reference circuit diameter** Q571: Definition of the diameter of the reference circuit. Specify in parameter Q573 whether the diameter references to the inscribed circle or the perimeter. Input range: 0 to 99999.9999

- **Workpiece blank diameter** Q222: Definition of the diameter of the workpiece blank. The workpiece blank diameter must be greater than the finished diameter. The TNC performs multiple stepovers if the difference between the workpiece blank diameter and reference circuit 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

- **Number of corners** Q572: Enter the number of corners of the polygon. The TNC will always equally divide the corners on the stud. Input range 3 to 30

- **Angle of rotation** Q224: Specify which angle is used to machine the first corner of the polygon. Input range: -360° to +360°
5.8 CIRCULAR STUD (cycle 258, DIN/ISO: G258)

- **Radius/Chamfer** Q220: Enter the value for the input form radius or chamfer. When entering a positive value between 0 and +99999.9999 the TNC creates a polygon with round corners. The radius refers to the value you entered. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.

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

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

- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (if you enter 0, climb milling is used for machining)

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of 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

### NC blocks

<table>
<thead>
<tr>
<th>8 CYCL DEF 258 POLYGON STUDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q573=1 ;REFERENCE CIRCUIT</td>
</tr>
<tr>
<td>Q571=50 ;REFERENCE CIRCUIT DIA.</td>
</tr>
<tr>
<td>Q222=120 ;WORKPIECE BLANK DIA.</td>
</tr>
<tr>
<td>Q572=10 ;NUMBER OF CORNERS</td>
</tr>
<tr>
<td>Q224=40 ;ANGLE OF ROTATION</td>
</tr>
<tr>
<td>Q220=2 ;RADIUS/CHAMFER</td>
</tr>
<tr>
<td>Q368=0 ;FINISHING ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q207=3000 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q351=1 ;MILLING TYPE</td>
</tr>
<tr>
<td>Q201=-18 ;DEPTH</td>
</tr>
<tr>
<td>Q202=10 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLUNGING</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>
</tbody>
</table>
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**

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

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

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

- **Machining operation (0/1/2)** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing

  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined

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

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

- **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**

```
Q215=0 ;MACHINING OPERATION
Q369=0 ;ALLOWANCE OF THE FLOOR
Q338=0 ;INFEED FOR FINISHING
Q385=500 ;FINISHING FEED RATE
9 L X+50 Y+50 R0 FMAX M3 M99
```
5.9 FACE MILLING (Cycle 233, DIN/ISO: G233)

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

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

1. From the current position, the TNC positions the tool at rapid traverse \( \text{FMAX} \) to the starting point 1 in the working plane: The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.

2. The TNC then positions the tool at rapid traverse \( \text{FMAX} \) to the set-up clearance in the spindle axis.

3. The tool then moves in the tool axis at the feed rate for milling \( \text{Q207} \) to the first plunging depth calculated by the TNC.
Strategies Q389=0 and Q389 =1

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

4 The TNC moves the tool to the end point 2 at the programmed feed rate for milling.

5 Then the TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and the safety clearance to the side.

6 The tool then returns at the feed rate for milling in the opposite direction.

7 The process is repeated until the programmed surface has been completed.

8 The TNC then positions the tool at rapid traverse FMAX back to the starting point 1.

9 If more than one infeed is required, the TNC moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.

10 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.

11 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.
Strategies Q389=2 and Q389=3

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

4 The tool subsequently advances to the end point 2 at the programmed feed rate for milling.

5 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at \( F_{\text{MAX}} \) directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the safety clearance to the side.

6 The tool then returns to the current infeed depth and moves in the direction of the next end point 2.

7 The multipass process is repeated until the programmed surface has been completed. At the end of the last path, the TNC positions the tool at rapid traverse \( F_{\text{MAX}} \) back to the starting point 1.

8 If more than one infeed is required, the TNC moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.

9 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.

10 At the end of the cycle, the tool is retracted at \( F_{\text{MAX}} \) to the 2nd set-up clearance.
Strategy Q389=4

4 The tool subsequently approaches the starting point of the milling path on a tangential arc at the programmed feed rate for milling.

5 The TNC machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged.

6 The process is repeated until the programmed surface has been completed. At the end of the last path, the TNC positions the tool at rapid traverse $F_{MAX}$ back to the starting point 1.

7 If more than one infeed is required, the TNC moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.

8 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at $F_{MAX}$ to the 2nd set-up clearance.

Limits

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

Pre-position the tool in the machining plane to the starting position with radius compensation R0. Keep in mind the machining direction. The TNC automatically pre-positions the tool in the tool axis. Note the 2nd set-up clearance Q204. Enter the 2nd set-up clearance in Q204 so that no collision with the workpiece or the fixtures can occur. If the starting point in the 3rd axis Q227 and the end point in the 3rd axis Q386 are entered as equal values, the TNC does not run the cycle (depth = 0 has been programmed).

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

- **Machining operation (0/1/2) Q215**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing

Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined.

- **Milling strategy (0 to 4) Q389**: Determine how the TNC should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface to be machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate outside the surface to be machined
  3: Line-by-line machining, retraction and stepover at the positioning feed rate at the edge of the surface to be machined
  4: Helical machining, uniform infeed from the outside toward the inside

- **Milling direction Q350**: Axis in the machining plane that defines the machining direction:
  1: Reference axis = machining direction
  2: Minor axis = machining direction

- **1st side length Q218** (incremental): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in the 1st axis. Input range 0 to 99999.9999

- **2nd side length Q219** (incremental value): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the starting point in the 2nd axis. Input range -99999.9999 to 99999.9999
## 5.9 FACE MILLING (Cycle 233)

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

### NC blocks

<table>
<thead>
<tr>
<th>8 CYCL DEF 233 FACE MILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=0 :MACHINING OPERATION</td>
</tr>
<tr>
<td>Q389=2 :MILLING STRATEGY</td>
</tr>
<tr>
<td>Q350=1 :MILLING DIRECTION</td>
</tr>
<tr>
<td>Q218=120 :1ST SIDE LENGTH</td>
</tr>
<tr>
<td>Q219=80 :2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q227=0 :STARTNG PNT 3RD AXIS</td>
</tr>
<tr>
<td>Q369=0.2 :ALLOWANCE FOR FLOOR</td>
</tr>
<tr>
<td>Q202=3 :MAX, PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q370=1 :TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q207=500 :FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q385=500 :FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q253=750 :F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q357=2 :CLEARANCE TO SIDE</td>
</tr>
<tr>
<td>Q200=2 :SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q204=50 :2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q347=0 :1ST LIMIT</td>
</tr>
<tr>
<td>Q348=0 :2ND LIMIT</td>
</tr>
<tr>
<td>Q349=0 :3RD LIMIT</td>
</tr>
<tr>
<td>Q220=2 :CORNER RADIUS</td>
</tr>
<tr>
<td>Q368=0 :ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q338=0 :INFEED FOR FINISHING</td>
</tr>
<tr>
<td>9 L X+0 Y+0 R0 FMAX M3 M99</td>
</tr>
</tbody>
</table>
2nd set-up clearance Q204 (incremental)
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.
Input range 0 to 99999.9999; alternatively PREDEF

1st limit Q347: Select the workpiece side on which the level surface is limited by a side wall (not possible for helical machining). Depending on the position of the side wall, the TNC limits the machining of the level surface to the respective coordinate of the starting point or to the side length: (not possible for helical machining):
Input 0: No limit
Input -1: Limit in the negative reference axis
Input +1: Limit in the positive reference axis
Input -2: Limit in the negative minor axis
Input +2: Limit in the positive minor axis

2nd limit Q348: See parameter 1st limit Q347

3rd limit Q349: See parameter 1st limit Q347

Corner radius Q220: Radius for corner at limits (Q347 to Q349). Input range 0 to 99999.9999

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

Infeed for finishing Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
## 5.10 Programming Examples

Example: Milling pockets, studs and slots

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGINN PGM C210 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 S3500</td>
<td>Call the tool for roughing/finishing</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 256 RECTANGULAR STUD</td>
<td>Define cycle for machining the contour outside</td>
</tr>
<tr>
<td></td>
<td>Q218=90 ;FIRST SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q424=100 ;WORKPC. BLANK SIDE 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q219=80 ;2ND SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q425=100 ;WORKPC. BLANK SIDE 2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q220=0 ;CORNER RADIUS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q368=0 ;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q224=0 ;ANGLE OF ROTATION</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q367=0 ;STUD POSITION</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q207=250 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q351=-1 ;CLIMB OR UP-CUT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q201=-30 ;DEPTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q202=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q206=250 ;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q200=2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q203=+0 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q204=20 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q370=1 ;TOOL PATH OVERLAP</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q437=0 ;APPROACH POSITION</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>L X+50 Y+50 R0 M3 M99</td>
<td>Call cycle for machining the contour outside</td>
</tr>
<tr>
<td>7</td>
<td>CYCL DEF 252 CIRCULAR POCKET</td>
<td>Define CIRCULAR POCKET MILLING cycle</td>
</tr>
<tr>
<td></td>
<td>Q215=0 ;MACHINING OPERATION</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q223=50 ;CIRCLE DIAMETER</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q368=0.2 ;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
</tbody>
</table>
### Programming Examples 5.10

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q351=+1</td>
<td>;CLIMB OR UP-CUT</td>
<td></td>
</tr>
<tr>
<td>Q201=-30</td>
<td>;DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q202=5</td>
<td>;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q369=0.1</td>
<td>;ALLOWANCE FOR FLOOR</td>
<td></td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td>Q338=5</td>
<td>;INFEED FOR FINISHING</td>
<td></td>
</tr>
<tr>
<td>Q200=2</td>
<td>;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q370=1</td>
<td>;TOOL PATH OVERLAP</td>
<td></td>
</tr>
<tr>
<td>Q366=1</td>
<td>;PLUNGE</td>
<td></td>
</tr>
<tr>
<td>Q385=750</td>
<td>;FINISHING FEED RATE</td>
<td></td>
</tr>
<tr>
<td>8 L X+50 Y+50 R0 FMAX M99</td>
<td>Call CIRCULAR POCKET MILLING cycle</td>
<td></td>
</tr>
<tr>
<td>9 L Z+250 R0 FMAX M6</td>
<td>Tool change</td>
<td></td>
</tr>
<tr>
<td>10 TOLL CALL 2 Z S5000</td>
<td>Call tool: slotting mill</td>
<td></td>
</tr>
<tr>
<td>11 CYCL DEF 254 CIRCULAR SLOT</td>
<td>Define SLOT cycle</td>
<td></td>
</tr>
<tr>
<td>Q215=0</td>
<td>;MACHINING OPERATION</td>
<td></td>
</tr>
<tr>
<td>Q219=8</td>
<td>;SLOT WIDTH</td>
<td></td>
</tr>
<tr>
<td>Q368=0.2</td>
<td>;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td>Q375=70</td>
<td>;PITCH CIRCLE DIA.</td>
<td></td>
</tr>
<tr>
<td>Q367=0</td>
<td>;REF. SLOT POSITION</td>
<td></td>
</tr>
<tr>
<td>Q216=-50</td>
<td>;CENTER IN 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td>Q217=-50</td>
<td>;CENTER IN 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td>Q376=-45</td>
<td>;STARTING ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q248=90</td>
<td>;ANGULAR LENGTH</td>
<td></td>
</tr>
<tr>
<td>Q378=180</td>
<td>;STEPPING ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q377=2</td>
<td>;NR OF REPETITIONS</td>
<td></td>
</tr>
<tr>
<td>Q207=500</td>
<td>;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>Q351=+1</td>
<td>;CLIMB OR UP-CUT</td>
<td></td>
</tr>
<tr>
<td>Q201=-20</td>
<td>;DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q202=5</td>
<td>;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q369=0.1</td>
<td>;ALLOWANCE FOR FLOOR</td>
<td></td>
</tr>
<tr>
<td>Q206=150</td>
<td>;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td>Q338=5</td>
<td>;INFEED FOR FINISHING</td>
<td></td>
</tr>
<tr>
<td>Q200=2</td>
<td>;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q203=+0</td>
<td>;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>Q204=50</td>
<td>;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q366=1</td>
<td>;PLUNGE</td>
<td></td>
</tr>
<tr>
<td>12 CYCL CALL FMAX M3</td>
<td>Call SLOT cycle</td>
<td></td>
</tr>
<tr>
<td>13 L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
<td></td>
</tr>
<tr>
<td>14 END PGM C210 MM</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fixed Cycles: Pattern Definitions
6.1 Fundamentals

Overview
The TNC provides two cycles for machining point patterns directly:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>220</td>
<td>POLAR PATTERN</td>
<td>187</td>
</tr>
<tr>
<td>221</td>
<td>CARTESIAN PATTERN</td>
<td>190</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", page 71) to develop point tables.
More regular point patterns are available with the PATTERN DEF function (see "PATTERN DEF pattern definition", page 64).

Cycle 200 DRILLING
Cycle 201 REAMING
Cycle 202 BORING
Cycle 203 UNIVERSAL DRILLING
Cycle 204 BACK BORING
Cycle 205 UNIVERSAL PECKING
Cycle 206 TAPPING NEW with a floating tap holder
Cycle 207 RIGID TAPPING without a floating tap holder NEW
Cycle 208 BORE MILLING
Cycle 209 TAPPING WITH CHIP BREAKING
Cycle 240 CENTERING
Cycle 251 RECTANGULAR POCKET
Cycle 252 CIRCULAR POCKET MILLING
Cycle 253 SLOT MILLING
Cycle 254 CIRCULAR SLOT (can only be combined with Cycle 221)
Cycle 256 RECTANGULAR STUD
Cycle 257 CIRCULAR STUD
Cycle 262 THREAD MILLING
Cycle 263 THREAD MILLING/COUNTERSINKING
Cycle 264 THREAD DRILLING/MILLING
Cycle 265 HELICAL THREAD DRILLING/MILLING
Cycle 267 OUTSIDE THREAD MILLING
6.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220)

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

Please note while programming:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.
If you combine Cycle 220 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and 2nd set-up clearance from Cycle 220 will be effective for the selected fixed cycle.
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.
Fixed Cycles: Pattern Definitions

6.2 POLAR PATTERN (Cycle 220)

Cycle parameters

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

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>53 CYCL DEF 220 POLAR PATTERN</td>
</tr>
<tr>
<td>Q216=+50 ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q217=+50 ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q244=80 ;PITCH CIRCLE DIA.</td>
</tr>
<tr>
<td>Q245=+0 ;STARTING ANGLE</td>
</tr>
<tr>
<td>Q246=+360;STOPPING ANGLE</td>
</tr>
<tr>
<td>Q247=+0 ;STEPPING ANGLE</td>
</tr>
<tr>
<td>Q241=8 ;NR OF REPETITIONS</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+30 ;SURFACE COORDINATE</td>
</tr>
</tbody>
</table>
- **2nd set-up clearance Q204 (incremental):** Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

- **Traversing to clearance height Q301:** Definition of how the touch probe is to move between machining operations:
  - 0: Move at set-up clearance between machining operations
  - 1: Move at 2nd set-up clearance between machining operations

- **Type of traverse? Line=0/Arc=1 Q365:** Definition of the path function with which the tool moves between machining operations:
  - 0: Move in a straight line between machining operations
  - 1: Move in a circular arc on the pitch circle diameter between machining operations

<table>
<thead>
<tr>
<th>Q204=50</th>
<th>;2ND SET-UP CLEARANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q301=1</td>
<td>;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q365=0</td>
<td>;TYPE OF TRAVERSE</td>
</tr>
</tbody>
</table>
Fixed Cycles: Pattern Definitions

6.3 LINEAR PATTERN (Cycle 221)

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

Please note while programming:

- Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.
- If you combine Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface, 2nd set-up clearance and rotational position that you defined in Cycle 221 will be effective for the selected fixed cycle.
- The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.
- If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.
Cycle parameters

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

**NC blocks**

<table>
<thead>
<tr>
<th>S4 CYCL DEF 221 CARTESIAN PATTERN</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q225=+15 ;STARTNG PNT 1ST AXIS</td>
</tr>
<tr>
<td>Q226=+15 ;STARTNG PNT 2ND AXIS</td>
</tr>
<tr>
<td>Q237=+10 ;SPACING IN 1ST AXIS</td>
</tr>
<tr>
<td>Q238=+8 ;SPACING IN 2ND AXIS</td>
</tr>
<tr>
<td>Q242=6 ;NUMBER OF COLUMNS</td>
</tr>
<tr>
<td>Q243=4 ;NUMBER OF LINES</td>
</tr>
<tr>
<td>Q224=+15 ;ANGLE OF ROTATION</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+30 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q301=1 ;MOVE TO CLEARANCE</td>
</tr>
</tbody>
</table>
### 6.4 Programming Examples

#### Example: Polar hole patterns

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 BEGIN PGM HOLEPAT MM</td>
<td>Definition of workpiece blank</td>
</tr>
<tr>
<td>1 BLK FORM 0.1 Z X+0 Y+0 Z-40</td>
<td>Definition of workpiece blank</td>
</tr>
<tr>
<td>2 BLK FORM 0.2 Y+100 Y=100 Z=0</td>
<td>Tool call</td>
</tr>
<tr>
<td>3 TOOL CALL 1 Z S3500</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>4 L Z+250 R0 FMAX M3</td>
<td>Cycle definition: drilling</td>
</tr>
<tr>
<td>5 CYCL DEF 200 DRILLING</td>
<td>Cycle definition: drilling</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q201=-15 ;DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q206=250 ;FEED RATE FOR PLUNGING</td>
<td></td>
</tr>
<tr>
<td>Q202=4 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q211=0 ;DWELL TIME AT TOP</td>
<td></td>
</tr>
<tr>
<td>Q203=+0 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>Q204=100 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q211=0.25 ;DWELL TIME AT BOTTOM</td>
<td></td>
</tr>
<tr>
<td>6 CYCL DEF 220 POLAR PATTERN</td>
<td>Define cycle for polar pattern 1, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220.</td>
</tr>
<tr>
<td>Q216=+30 ;CENTER IN 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td>Q217=+70 ;CENTER IN 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td>Q244=50 ;PITCH CIRCLE DIA.</td>
<td></td>
</tr>
<tr>
<td>Q245=+0 ;STARTING ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q246=+360 ;STOPPING ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q247=+0 ;STEPPING ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q241=10 ;NR OF REPETITIONS</td>
<td></td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q203=+0 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td>Q204=100 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Line</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
</tr>
<tr>
<td>301=1</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>365=0</td>
<td>TYPE OF TRAVERSE</td>
</tr>
<tr>
<td>7 CYCL DEF 220 POLAR PATTERN</td>
<td>Define cycle for polar pattern 2, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220.</td>
</tr>
<tr>
<td>216=+90</td>
<td>CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>217=+25</td>
<td>CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>244=70</td>
<td>PITCH CIRCLE DIA.</td>
</tr>
<tr>
<td>245=+90</td>
<td>STARTING ANGLE</td>
</tr>
<tr>
<td>246=+360</td>
<td>STOPPING ANGLE</td>
</tr>
<tr>
<td>247=+30</td>
<td>STEPPING ANGLE</td>
</tr>
<tr>
<td>241=5</td>
<td>NR OF REPETITIONS</td>
</tr>
<tr>
<td>200=2</td>
<td>SET-UP CLEARANCE</td>
</tr>
<tr>
<td>203=+0</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td>204=100</td>
<td>2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>301=1</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>365=0</td>
<td>TYPE OF TRAVERSE</td>
</tr>
<tr>
<td>8 L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>9 END PGM HOLEPAT MM</td>
<td></td>
</tr>
</tbody>
</table>
7

Fixed Cycles: Contour Pocket
7 Fixed Cycles: Contour Pocket

7.1 SL Cycles

7.1 SL Cycles

Fundamentals

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

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

SL cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always run a graphical program test before machining! This is a simple way of finding out whether the TNC-calculated program will provide the desired results.

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

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.

- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR.

- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL.

- The subprograms must not contain spindle axis coordinates.

- Always program both axes in the first block of the subprogram.

- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms.

Program structure: Machining with SL cycles

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

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

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

Overview

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>CONTOUR GEOMETRY</td>
<td>198</td>
</tr>
<tr>
<td>20</td>
<td>CONTOUR DATA</td>
<td>203</td>
</tr>
<tr>
<td>21</td>
<td>PILOT DRILLING</td>
<td>205</td>
</tr>
<tr>
<td>22</td>
<td>ROUGH-OUT</td>
<td>207</td>
</tr>
<tr>
<td>23</td>
<td>FLOOR FINISHING</td>
<td>211</td>
</tr>
<tr>
<td>24</td>
<td>SIDE FINISHING</td>
<td>213</td>
</tr>
</tbody>
</table>

Enhanced cycles:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>CONTOUR TRAIN</td>
<td>216</td>
</tr>
<tr>
<td>270</td>
<td>CONTOUR TRAIN DATA</td>
<td>218</td>
</tr>
</tbody>
</table>
7 Fixed Cycles: Contour Pocket

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

Please note while programming:
All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY.

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

Cycle parameters

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

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

Subprograms: overlapping pockets

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

Pockets A and B overlap. The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed). The pockets are programmed as full circles.

Subprogram 1: Pocket A

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

Subprogram 2: Pocket B

56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
Fixed Cycles: Contour Pocket

7.3 Superimposed contours

Area of inclusion
Both surfaces A and B are to be machined, including the overlapping area:
- The surfaces A and B must be pockets.
- The first pocket (in Cycle 14) must start outside the second pocket.

Surface A:

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

Surface B:

56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
Area of exclusion
Surface A is to be machined without the portion overlapped by B:
- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.

Surface A:
- LBL 1
- L X+10 Y+50 RR
- CC X+35 Y+50
- C X+10 Y+50 DR-
- LBL 0

Surface B:
- LBL 2
- L X+40 Y+50 RL
- CC X+65 Y+50
- C X+40 Y+50 DR-
- 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 unmaechined.)

- 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
Please note while programming:
Machining data for the subprograms describing the subcontours are entered in Cycle 20.

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

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

Cycle parameters

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

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

NC blocks

<table>
<thead>
<tr>
<th>57 CYCL DEF 20 CONTOUR DATA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q1=-20 ;MILLING DEPTH</td>
</tr>
<tr>
<td>Q2=1 ;TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q3=+0.2 ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q4=+0.1 ;ALLOWANCE FOR FLOOR</td>
</tr>
<tr>
<td>Q5=+30 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q6=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q7=+80 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q8=0.5 ;ROUNDING RADIUS</td>
</tr>
<tr>
<td>Q9=+1 ;DIRECTION OF ROTATION</td>
</tr>
</tbody>
</table>
7.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121)

Cycle run

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

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

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

Cycle run:

1. The TNC first positions the tool in the plane (the position results from the contour you have defined with Cycle 14 or SEL CONTOUR, and from the rough-out tool data).

2. The tool then moves at rapid traverse \( F_{MAX} \) to the set-up clearance. (Define the set-up clearance in Cycle 20 CONTOUR DATA).

3. The tool drills from the current position to the first plunging depth at the programmed feed rate \( F \).

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

5. The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: \( t = 0.6 \) mm
   - At a total hole depth exceeding 30 mm: \( t = \) hole depth / 50
   - Maximum advanced stop distance: 7 mm

6. The tool then advances with another infeed at the programmed feed rate \( F \).

7. The TNC repeats this process (1 to 4) until the programmed total hole depth is reached. The finishing allowance for floor is taken into account.

8. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.
Fixed Cycles: Contour Pocket

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

Please note while programming:

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

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

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

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

Cycle parameters

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

NC blocks

<table>
<thead>
<tr>
<th>58 CYCL DEF 21 PILOT DRILLING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q10=+5 :PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q11=100 :FEED RATE FOR PLUNGNG</td>
</tr>
<tr>
<td>Q13=1  :ROUGH-OUT TOOL</td>
</tr>
</tbody>
</table>
7.6 ROUGHING (Cycle 22, DIN/ISO: G122)

Cycle run

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

Before calling Cycle 22 you need to program further cycles:
- Cycle 14 CONTOUR GEOMETRY or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary

Cycle run

1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.

2. In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate.

3. First the island contours (C and D in the figure at right) are rough-milled until the pocket contour (A, B) is approached.

4. In the next step the TNC moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached.

5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the parameter ConfigDatum, CfgGeoCycle, posAfterContPocket.
Please note while programming:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21. You define the plunging behavior of Cycle 22 with parameter Q19 and with the tool table in the ANGLE and LCUTS columns:

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

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

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

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

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

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

### NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>S9 CYCL DEF 22 ROUGH-OUT</td>
</tr>
<tr>
<td>Q10=+5 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q11=100 ;FEED RATE FOR PLUNGNG</td>
</tr>
<tr>
<td>Q12=750 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q18=1 ;COARSE ROUGHING TOOL</td>
</tr>
<tr>
<td>Q19=150 ;RECIPROCATION FEED RATE</td>
</tr>
<tr>
<td>Q208=9999 ;RETRACTION FEED RATE</td>
</tr>
<tr>
<td>Q401=80 ;FEED RATE REDUCTION</td>
</tr>
<tr>
<td>Q404=0 ;FINE ROUGH STRATEGY</td>
</tr>
</tbody>
</table>
7 Fixed Cycles: Contour Pocket

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

- **Feed rate factor in % Q401**: Percentage factor by which the TNC reduces the machining feed rate (Q12) as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (Q2) specified in Cycle 20. The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machining time should be reduced in total. Input range 0.0001 to 100.0000.

- **Fine rough strategy Q404**: Specify the fine roughing behavior of the TNC if the radius of the fine roughing tool is greater than half the diameter of the rough-out tool:
  - Q404=0: The TNC moves the tool between the areas to be fine roughed at the current depth along the contour.
  - Q404=1: The TNC retracts the tool to the set-up clearance between the areas to be fine roughed and then moves to the starting point for the next area to be roughed out.
7.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123)

Cycle run

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

Before calling Cycle 23 you need to program further cycles:
- Cycle 14 CONTOUR GEOMETRY or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary
- Cycle 22 ROUGHING, if necessary

Cycle run

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

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

Please note while programming:

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

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

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

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

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

Cycle parameters

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

NC blocks

<table>
<thead>
<tr>
<th>60 CYCL DEF 23 FLOOR FINISHING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q11=100 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q12=350 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q208=9999;RETRACTION FEED RATE</td>
</tr>
</tbody>
</table>
7.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124)

Cycle run

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

Before calling Cycle 24 you need to program further cycles:
- Cycle 14 CONTOUR GEOMETRY or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary
- Cycle 22 ROUGHING, if necessary

Cycle run

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

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

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

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

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

You can use Cycle 24 also for contour milling. Then you must:
- define the contour to be milled as a single island (without pocket limit), and
- enter the finishing allowance (Q3) in Cycle 20 to be greater than the sum of the finishing allowance Q14 + radius of the tool being used.

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

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

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

Danger of collision!

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

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

- **Direction of rotation** Q9: Machining direction:
  - +1: Rotation counterclockwise
  - -1: Rotation clockwise

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

- **Feed rate for plunging** Q11: Traversing speed of the tool when plunging into the workpiece in mm/min. Input range 0 to 99999.9999 alternatively FAUTO, FU, FZ

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

- **Finishing allowance for side** Q14 (incremental): The allowance for side Q14 is left over after finishing. (This allowance must be smaller than the allowance in Cycle 20.) Input range -99999.9999 to 99999.9999

---

**NC blocks**

| 61 CYCL DEF 24 SIDE FINISHING |
|---|---|
| Q9=+1 | DIRECTION OF ROTATION |
| Q10=+5 | PLUNGING DEPTH |
| Q11=100 | FEED RATE FOR PLUNGING |
| Q12=350 | FEED RATE FOR MILLING |
| Q14=-0 | ALLOWANCE FOR SIDE |
Cycle run
In conjunction with Cycle 14 CONTOUR GEOMETRY, this cycle facilitates the machining of open and closed contours. Cycle 25 CONTOUR TRAIN offers considerable advantages over machining a contour using positioning blocks:

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

Please note while programming:

- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The TNC takes only the first label of Cycle 14 CONTOUR GEOMETRY into account.
- The sub program allows no APPr- or DEP motions.
- When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- Cycle 20 CONTOUR DATA is not required.
- If M110 is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.
Danger of collision!
To avoid collisions,
  - Do not program positions in incremental dimensions immediately after Cycle 25 since they are referenced to the position of the tool at the end of the cycle.
  - Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.

**Cycle parameters**

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

**NC blocks**

<table>
<thead>
<tr>
<th>NC block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>62 CYCL DEF 25 CONTOUR TRAIN</td>
<td></td>
</tr>
<tr>
<td>Q1=-20</td>
<td>MILLING DEPTH</td>
</tr>
<tr>
<td>Q3=+0</td>
<td>ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q5=+0</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q7=+50</td>
<td>CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q10=+5</td>
<td>PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q11=100</td>
<td>FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q12=350</td>
<td>FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q15=-1</td>
<td>CLIMB OR UP-CUT</td>
</tr>
</tbody>
</table>
7 Fixed Cycles: Contour Pocket

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

Please note while programming:
You can use this cycle to specify various properties of Cycle 25 CONTOUR TRAIN.

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

Cycle parameters

- **Type of approach/departure (1/2/3)** Q390:
  Definition of the type of approach and departure:
  Q390=1: Approach the contour tangentially on a circular path
  Q390=2: Approach the contour tangentially on a straight line
  Q390=3: Approach the contour at a right angle

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

- **Approach/departure radius** Q392: Only in effect if tangential approach on a circular path is selected (Q390=1). Radius of the approach/departure arc. Input range 0 to 99999.9999

- **Center angle** Q393: Only in effect if tangential approach on a circular path is selected (Q390=1). Angular length of the approach arc. Input range 0 to 99999.9999

- **Distance to auxiliary point** Q394: Only in effect if tangential approach on a straight line or right-angle approach is selected (Q390=2 or Q390=3). Distance to the auxiliary point from which the TNC is to approach the contour. Input range 0 to 99999.9999

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>62 CYCL DEF 270 CONTOUR TRAIN DATA</td>
</tr>
<tr>
<td>Q390=1 ;TYPE OF APPROACH</td>
</tr>
<tr>
<td>Q391=1 ;RADIUS COMPENSATION</td>
</tr>
<tr>
<td>Q392=3 ;RADIUS</td>
</tr>
<tr>
<td>Q393=+45 ;CENTER ANGLE</td>
</tr>
<tr>
<td>Q394=+2 ;DISTANCE</td>
</tr>
</tbody>
</table>
7.11 TROCHOIDAL SLOT (Cycle 275, DIN ISO G275)

Cycle run

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

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

Depending on the cycle parameters you select, the following machining alternatives are available:
- Complete machining: Roughing, side finishing
- Only roughing
- Only side finishing

Roughing with closed slots

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

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

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

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

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

Finishing with closed slots

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

Program structure: Machining with SL cycles

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

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

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

**Finishing with open slots**

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

**Please note while programming:**

- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
  - When using Cycle 275 TROCHOIDAL SLOT, you can define only one contour subprogram in Cycle 14 CONTOUR GEOMETRY.
  - Define the center line of the slot with all available path functions in the contour subprogram.
  - The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
  - The TNC does not need Cycle 20 CONTOUR DATA in conjunction with Cycle 275.
  - The starting point of a closed slot must not be located in a contour corner.

**Danger of collision!**

To avoid collisions,

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

- **Machining operation (0/1/2) Q215**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined

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

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

- **Infeed per rev. Q436 absolute**: Value by which the TNC moves the tool in the machining direction per revolution. Input range 0 to 99999.9999

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

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

- **Climb or up-cut Q351**: Type of milling operation with M3
  +1 = climb
  -1 = up-cut
  **PREDEF**: The TNC uses the value from the GLOBAL DEF block (if you enter 0, climb milling is used for machining)

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

7.11 TROCHOIDAL SLOT (Cycle 275, DIN ISO G275)

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

### NC blocks

<table>
<thead>
<tr>
<th>8 CYCL DEF 275 TROCHOIDAL SLOT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=0 ; MACHINING OPERATION</td>
</tr>
<tr>
<td>Q219=12 ; SLOT WIDTH</td>
</tr>
<tr>
<td>Q368=0.2 ; ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q436=2 ; INFEED PER REV.</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>Q206=150 ; FEED RATE FOR PLAGNG</td>
</tr>
<tr>
<td>Q338=5 ; INFEED FOR FINISHING</td>
</tr>
<tr>
<td>Q385=500 ; FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q200=2 ; SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q202=5 ; PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q203=+0 ; SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ; 2ND SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q366=2 ; PLUNGE</td>
</tr>
</tbody>
</table>

| 9 CYCL CALL FMAX M3             |

---

222

HEIDENHAIN | TNC 640 | User's manual for cycle programming | 9/2015
7.12 Programming Examples

Example: Roughing-out and fine-roughing a pocket

```
0 BEGIN PGM C20 MM
1 BLK FORM 0.1 Z X-10 Y-10 Z-40
2 BLK FORM 0.2 X+100 Y+100 Z+0
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 ;SET-UP CLEARANCE
   Q7=+100 ;CLEARANCE HEIGHT
   Q8=0.1 ;ROUNDING RADIUS
   Q9=-1 ;DIRECTION
8 CYCL DEF 22 ROUGH-OUT
   Q10=5 ;PLUNGING DEPTH
   Q11=100 ;FEED RATE FOR PLNGNG
   Q12=350 ;FEED RATE FOR ROUGH-OUT
   Q18=0 ;COARSE ROUGHING TOOL
   Q19=150 ;RECIROCATION FEED RATE
   Q208=30000 ;RETRACTION FEED RATE
9 CYCL CALL M3
10 L Z+250 R0 FMAX M6
```

Cycle definition: Coarse roughing
Cycle call: Coarse roughing
Tool change
## Fixed Cycles: Contour Pocket

### 7.12 Programming Examples

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

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM C21 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 S2500</td>
<td>Tool call: Drill, diameter 12</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 /2 /3 /4</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>CYCL DEF 20 CONTOUR DATA</td>
<td>Define general machining parameters</td>
</tr>
<tr>
<td></td>
<td>Q1=-20</td>
<td>MILLING DEPTH</td>
</tr>
<tr>
<td></td>
<td>Q2=1</td>
<td>TOOL PATH OVERLAP</td>
</tr>
<tr>
<td></td>
<td>Q3=0.5</td>
<td>ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td></td>
<td>Q4=0.5</td>
<td>ALLOWANCE FOR FLOOR</td>
</tr>
<tr>
<td></td>
<td>Q5=0</td>
<td>SURFACE COORDINATE</td>
</tr>
<tr>
<td></td>
<td>Q6=2</td>
<td>SET-UP CLEARANCE</td>
</tr>
<tr>
<td></td>
<td>Q7=100</td>
<td>CLEARANCE HEIGHT</td>
</tr>
<tr>
<td></td>
<td>Q8=0.1</td>
<td>ROUNding RADIUS</td>
</tr>
<tr>
<td></td>
<td>Q9=-1</td>
<td>DIRECTION</td>
</tr>
<tr>
<td>8</td>
<td>CYCL DEF 21 PILOT DRILLING</td>
<td>Cycle definition: Pilot drilling</td>
</tr>
<tr>
<td></td>
<td>Q10=5</td>
<td>PLUNGING DEPTH</td>
</tr>
<tr>
<td></td>
<td>Q11=250</td>
<td>FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td></td>
<td>Q13=2</td>
<td>ROUGH-OUT TOOL</td>
</tr>
<tr>
<td>9</td>
<td>CYCL CALL M3</td>
<td>Cycle call: Pilot drilling</td>
</tr>
<tr>
<td>10</td>
<td>L +250 R0 FMAX M6</td>
<td>Tool change</td>
</tr>
<tr>
<td>11</td>
<td>TOOL CALL 2 Z S3000</td>
<td>Call the tool for roughing/finishing, diameter 12</td>
</tr>
<tr>
<td>12</td>
<td>CYCL DEF 22 ROUGH-OUT</td>
<td>Cycle definition: Rough-out</td>
</tr>
<tr>
<td></td>
<td>Q10=5</td>
<td>PLUNGING DEPTH</td>
</tr>
<tr>
<td></td>
<td>Q11=100</td>
<td>FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td></td>
<td>Q12=350</td>
<td>FEED RATE FOR ROUGH-OUT</td>
</tr>
</tbody>
</table>
Q18 = 0 ;COARSE ROUGHING TOOL
Q19 = 150 ;RECIPROCATION FEED RATE
Q208 = 30000 ;RETRACTION FEED RATE

13 CYCL CALL M3
Cycle call: Rough-out

14 CYCL DEF 23 FLOOR FINISHING
Cycle definition: Floor finishing
Q11 = 100 ;FEED RATE FOR PLNGNG
Q12 = 200 ;FEED RATE FOR MILLING
Q208 = 30000 ;RETRACTION FEED RATE

15 CYCL CALL
Cycle call: Floor finishing

16 CYCL DEF 24 SIDE FINISHING
Cycle definition: Side finishing
Q9 = +1 ;DIRECTION OF ROTATION
Q10 = 5 ;PLUNGING DEPTH
Q11 = 100 ;FEED RATE FOR PLNGNG
Q12 = 400 ;FEED RATE FOR MILLING
Q14 = +0 ;ALLOWANCE FOR SIDE

17 CYCL CALL
Cycle call: Side finishing

18 L Z+250 R0 FMAX M2
Retract the tool, end program

19 LBL 1
Contour subprogram 1: left pocket

20 CC X+35 Y+50
21 L X+10 Y+50 RR
22 C X+10 DR-

23 LBL 0

24 LBL 2
Contour subprogram 2: right pocket

25 CC X+65 Y+50
26 L X+90 Y+50 RR
27 C X+90 DR-

28 LBL 0

29 LBL 3
Contour subprogram 3: square left island

30 L X+27 Y+50 RL
31 L Y+58
32 L X+43
33 L Y+42
34 L X+27

35 LBL 0

36 LBL 4
Contour subprogram 4: triangular right island

37 L X+65 Y+42 RL
38 L X+57
39 L X+65 Y+58
40 L X+73 Y+42
41 LBL 0

42 END PGM C21 MM
Example: Contour train

<table>
<thead>
<tr>
<th>Line</th>
<th>Instruction</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></td>
<td>Q1=-20 ;MILLING DEPTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q3=+0 ;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q5=+0 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q7=+250 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q10=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q11=100 ;FEED RATE FOR PLUNGING</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q12=200 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td></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 the tool, end program</td>
</tr>
<tr>
<td>10</td>
<td>LBL 1</td>
<td>Contour subprogram</td>
</tr>
<tr>
<td>11</td>
<td>L X+0 Y+15 RL</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>L X+5 Y+20</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>CT X+5 Y+75</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>L Y+95</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>RND R7.5</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>L X+50</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>RND R7.5</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>L X+100 Y+80</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>LBL 0</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>END PGM C25 MM</td>
<td></td>
</tr>
</tbody>
</table>
Fixed Cycles: Cylindrical Surface
8 Fixed Cycles: Cylindrical Surface

8.1 Fundamentals

8.1 Fundamentals

Overview of cylindrical surface cycles

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>27</td>
<td>27 CYLINDER SURFACE</td>
<td>231</td>
</tr>
<tr>
<td>28</td>
<td>28 CYLINDER SURFACE slot milling</td>
<td>234</td>
</tr>
<tr>
<td>29</td>
<td>29 CYLINDER SURFACE ridge milling</td>
<td>237</td>
</tr>
<tr>
<td>39</td>
<td>39 CYLINDER SURFACE Contour</td>
<td>240</td>
</tr>
</tbody>
</table>
8.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, software option 1)

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

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

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

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

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

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

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

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

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

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

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

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

This cycle can also be used in a tilted working plane. The set-up clearance must be greater than the tool radius.

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

When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, software option 1) 8.2

Cycle parameters

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

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

- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999

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

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

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

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

- **Dimension type? deg=0 MM/INCH=1** Q17: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

**NC blocks**

<table>
<thead>
<tr>
<th>63 CYCL DEF 27 CYLINDER SURFACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q1=-8;MILLING DEPTH</td>
</tr>
<tr>
<td>Q3=-0;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q6=-0;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q10=-3;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q11=100;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q12=350;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q16=25;RADIUS</td>
</tr>
<tr>
<td>Q17=0;DIMENSION TYPE</td>
</tr>
</tbody>
</table>
Fixed Cycles: Cylindrical Surface

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

Cycle run

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

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

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

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

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

- Define the approaching behavior in ConfigDatum, CfgGeoCycle, apprDepCylWall
  - CircleTangential: Tangential approach and departure
  - LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

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

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

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

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

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

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

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

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

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

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

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

- **Milling depth** $Q_1$ (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** $Q_3$ (incremental): Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. Input range -99999.9999 to 99999.9999
- **Set-up clearance** $Q_6$ (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999
- **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
- **Cylinder radius** $Q_{16}$: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type?** $deg=0$ MM/INCH=1 $Q_{17}$: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- **Slot width** $Q_{20}$: Width of the slot to be machined. Input range -99999.9999 to 99999.9999
- **Tolerance** $Q_{21}$: If you use a tool smaller than the programmed slot width $Q_{20}$, 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 $Q_{21}$, 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 $Q_{21}$ you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the remachining takes. Input range for tolerance 0.0001 to 9.9999

### Recommendation

Use a tolerance of 0.02 mm.

### Function inactive

Enter 0 (default setting)

### NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>63 CYCL DEF 28 CYLINDER SURFACE</td>
</tr>
<tr>
<td>$Q_1=8$ ;MILLING DEPTH</td>
</tr>
<tr>
<td>$Q_3=+0$ ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>$Q_6=+0$ ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>$Q_{10}=+3$ ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>$Q_{11}=100$ ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>$Q_{12}=350$ ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>$Q_{16}=25$ ;RADIUS</td>
</tr>
<tr>
<td>$Q_{17}=0$ ;DIMENSION TYPE</td>
</tr>
<tr>
<td>$Q_{20}=12$ ;SLOT WIDTH</td>
</tr>
<tr>
<td>$Q_{21}=0$ ;TOLERANCE</td>
</tr>
</tbody>
</table>
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.
8.4 CYLINDER SURFACE Ridge milling (Cycle 29, DIN/ISO: G129, software option 1)

Please note while programming:

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

- In the first NC block of the contour program, always program both cylinder surface coordinates. The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

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

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

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

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

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

- In the parameter CfgGeoCycle, displaySpindleErr, on/off, define whether the TNC should output an error message (on) or not (off) if spindle rotation is not active when the cycle is called. The function needs to be adapted by your machine manufacturer.
**CYLINDER SURFACE Ridge milling (Cycle 29, DIN/ISO: G129, software option 1)**

**Cycle parameters**

- **Milling depth** $Q_1$ (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** $Q_3$ (incremental): Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. Input range -99999.9999 to 99999.9999
- **Set-up clearance** $Q_6$ (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999
- **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
- **Cylinder radius** $Q_{16}$: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? deg=0 MM/INCH=1** $Q_{17}$: The coordinates for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- **Ridge width** $Q_{20}$: Width of the ridge to be machined. Input range -99999.9999 to 99999.9999

**NC blocks**

<table>
<thead>
<tr>
<th>63 CYCL DEF 29 CYLINDER SURFACE RIDGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q1=-8 ;MILLING DEPTH</td>
</tr>
<tr>
<td>Q3=+0 ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q6=+0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q10=+3 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q11=100 ;FEED RATE FOR PLUNGNG</td>
</tr>
<tr>
<td>Q12=350 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q16=25 ;RADIUS</td>
</tr>
<tr>
<td>Q17=0 ;DIMENSION TYPE</td>
</tr>
<tr>
<td>Q20=12 ;RIDGE WIDTH</td>
</tr>
</tbody>
</table>
Fixed Cycles: Cylindrical Surface

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

Cycle run

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

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

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

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

1 The TNC positions the tool over the starting point of machining.
   The TNC locates the starting point next to the first point defined in the contour subprogram, offset by the tool diameter.

2 The TNC then moves the tool to the first plunging depth.
   The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate Q12. A finishing allowance programmed for the side is taken into account. (The approaching behavior depends on the parameter ConfigDatum, CfgGeoCycle, apprDepCylWall.)

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

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

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

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

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

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

- The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Ensure that the tool has enough space laterally for contour approach and departure.
- The cylinder must be set up centered on the rotary table. Set the reference point to the center of the rotary table.
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.
- The set-up clearance must be greater than the tool radius.
- The machining time can increase if the contour consists of many non-tangential contour elements.
- When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
- Define the approaching behavior in ConfigDatum,CfgGeoCycle, apprDepCylWall
  - CircleTangential: Tangential approach and departure
  - LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

Danger of collision!

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

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

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>63 CYCL DEF 39 CYL. SURFACE CONTOUR</td>
</tr>
<tr>
<td>Q1=-8 ;MILLING DEPTH</td>
</tr>
<tr>
<td>Q3=-0 ;ALLOWANCE FOR SIDE</td>
</tr>
<tr>
<td>Q6=+0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q10=-3 ;PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q11=100 ;FEED RATE FOR PLNGNG</td>
</tr>
<tr>
<td>Q12=350 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q16=25 ;RADIUS</td>
</tr>
<tr>
<td>Q17=0 ;DIMENSION TYPE</td>
</tr>
</tbody>
</table>
# Programming Examples 8.6

## Example: Cylinder surface with Cycle 27

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

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

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>RND R7.5</td>
</tr>
<tr>
<td>22</td>
<td>L X+40 Y+20</td>
</tr>
<tr>
<td>23</td>
<td>LBL 0</td>
</tr>
<tr>
<td>24</td>
<td>END PGM C27 MM</td>
</tr>
</tbody>
</table>
Example: Cylinder surface with Cycle 28

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

```
0 BEGIN PGM C28 MM
1 TOOL CALL 1 Z S2000  
2 L Z+250 R0 FMAX  
3 L X+50 Y+0 R0 FMAX  
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN FMAX  
5 CYCL DEF 14.0 CONTOUR GEOMETRY  
6 CYCL DEF 14.1 CONTOUR LABEL 1  
7 CYCL DEF 28 CYLINDER SURFACE  
  Q1=-7 ;MILLING DEPTH  
  Q3=+0 ;ALLOWANCE FOR SIDE  
  Q6=2 ;SET-UP CLEARANCE  
  Q10=-4 ;PLUNGING DEPTH  
  Q11=100 ;FEED RATE FOR PLNGNG  
  Q12=250 ;FEED RATE FOR MILLING  
  Q16=25 ;RADIUS  
  Q17=1 ;DIMENSION TYPE  
  Q20=10 ;SLOT WIDTH  
  Q21=0.02 ;TOLERANCE  
8 L C+0 R0 FMAX M3 M99  
9 L Z+250 R0 FMAX  
10 PLANE RESET TURN FMAX  
11 M2  
12 LBL 1  
13 L X+60 Y+0 RL  
14 L Y-35  
15 L X+40 Y-52.5  
16 L Y-70  
17 LBL 0  
18 END PGM C28 MM
```
Fixed Cycles: Contour Pocket with Contour Formula
9.1 SL cycles with complex contour formula

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

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

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

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

---

Program structure: Machining with SL cycles and complex contour formula

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

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

Characteristics of the fixed cycles

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

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

Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN PGM MODEL MM
1 DECLARE CONTOUR QC1 = "CIRCLE1"
2 DECLARE CONTOUR QC2 = "CIRCLEY" DEPTH15
3 DECLARE CONTOUR QC3 = "TRIANGLE" DEPTH10
4 DECLARE CONTOUR QC4 = "SQUARE" DEPTH5
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM MODEL MM
0 BEGIN PGM CIRCLE 1 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM CIRCLE 1 MM
0 BEGIN PGM CIRCLE31XY MM
... ...
4 END PGM CIRCLE 1 MM
0 BEGIN PGM CIRCLE31XY MM
```
Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

Selecting a program with contour definitions

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

- Show the soft-key row with special functions
- Select the menu for functions for contour and point machining
- Press the SEL CONTOUR soft key
- Enter the full name of the program with the contour definition and confirm with the END key

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

Defining contour descriptions

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

- 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
- Enter the full name of the program with the contour description and confirm with the END key, or if desired,
- Define a separate depth for the selected contour

With the entered contour designators QC you can include the various contours in the contour formula. If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).
Entering a complex contour formula

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

- 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>Soft key</th>
<th>Mathematical function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Cut with" /></td>
<td>e.g. QC10 = QC1 &amp; QC5</td>
</tr>
<tr>
<td><img src="image" alt="Joined with" /></td>
<td>e.g. QC25 = QC7</td>
</tr>
<tr>
<td><img src="image" alt="Joined with, but without cut" /></td>
<td>e.g. QC12 = QC5 ^ QC25</td>
</tr>
<tr>
<td><img src="image" alt="Without" /></td>
<td>e.g. QC25 = QC1 \ QC2</td>
</tr>
<tr>
<td><img src="image" alt="Parenthesis open" /></td>
<td>e.g. QC12 = QC1 * (QC2 + QC3)</td>
</tr>
<tr>
<td><img src="image" alt="Parenthesis closed" /></td>
<td>e.g. QC12 = QC1 * (QC2 + QC3)</td>
</tr>
</tbody>
</table>

Define single contour
e.g. QC12 = QC1
**Superimposed contours**

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

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

**Subprograms: overlapping pockets**

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

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

The pockets are programmed as full circles.

**Contour description program 1: pocket A**

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

**Contour description program 2: pocket B**

```
0 BEGIN PGM POCKET_B MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET_B MM
```
Area of inclusion
Both areas A and B are to be machined, including the overlapping area:
- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.

Contour definition program:

```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 exclusion
Area A is to be machined without the portion overlapped by B:
- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the area B is subtracted from the area A with the without function.

Contour definition program:

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

Contour definition program:

```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", page 197).
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
   Q6=2 ; SET-UP CLEARANCE
   Q7=+100 ; CLEARANCE HEIGHT
   Q8=0.1 ; Rounding radius
   Q9=-1 ; DIRECTION
```
### Fixed Cycles: Contour Pocket with Contour Formula

#### 9.1 SL cycles with complex contour formula

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 CYCL DEF 22 ROUGH-OUT</td>
<td>Cycle definition: Rough-out</td>
</tr>
<tr>
<td>Q10=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q11=100 ;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td>Q12=350 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>Q18=0 ;COARSE ROUGHING TOOL</td>
<td></td>
</tr>
<tr>
<td>Q19=150 ;RECIPIROCATION FEED RATE</td>
<td></td>
</tr>
<tr>
<td>Q401=100 ;FEED RATE FACTOR</td>
<td></td>
</tr>
<tr>
<td>Q404=0 ;FINE ROUGH STRATEGY</td>
<td></td>
</tr>
<tr>
<td>10 CYCL CALL M3</td>
<td>Cycle call: Rough-out</td>
</tr>
<tr>
<td>11 TOOL CALL 2 Z S5000</td>
<td>Tool call of finishing cutter</td>
</tr>
<tr>
<td>12 CYCL DEF 23 FLOOR FINISHING</td>
<td>Cycle definition: Floor finishing</td>
</tr>
<tr>
<td>Q11=100 ;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td>Q12=200 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL M3</td>
<td>Cycle call: Floor finishing</td>
</tr>
<tr>
<td>14 CYCL DEF 24 SIDE FINISHING</td>
<td>Cycle definition: Side finishing</td>
</tr>
<tr>
<td>Q9=+1 ;DIRECTION OF ROTATION</td>
<td></td>
</tr>
<tr>
<td>Q10=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q11=100 ;FEED RATE FOR PLNGNG</td>
<td></td>
</tr>
<tr>
<td>Q12=400 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td>Q14=0 ;ALLOWANCE FOR SIDE</td>
<td></td>
</tr>
<tr>
<td>15 CYCL CALL M3</td>
<td>Cycle call: Side finishing</td>
</tr>
<tr>
<td>16 L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>17 END PGM CONTOUR MM</td>
<td></td>
</tr>
</tbody>
</table>

**Contour definition program with contour formula:**

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 BEGIN PGM MODEL MM</td>
<td>Contour definition program</td>
</tr>
<tr>
<td>1 DECLARE CONTOUR QC1 = &quot;CIRCLE1&quot;</td>
<td>Definition of the contour designator for the program &quot;CIRCLE1&quot;</td>
</tr>
<tr>
<td>2 FN 0: Q1 =+35</td>
<td>Assignment of values for parameters used in PGM &quot;CIRCLE31XY&quot;</td>
</tr>
<tr>
<td>3 FN 0: Q2 =+50</td>
<td></td>
</tr>
<tr>
<td>4 FN 0: Q3 =+25</td>
<td></td>
</tr>
<tr>
<td>5 DECLARE CONTOUR QC2 = &quot;CIRCLE31XY&quot;</td>
<td>Definition of the contour designator for the program &quot;CIRCLE31XY&quot;</td>
</tr>
<tr>
<td>6 DECLARE CONTOUR QC3 = &quot;TRIANGLE&quot;</td>
<td>Definition of the contour designator for the program &quot;TRIANGLE&quot;</td>
</tr>
<tr>
<td>7 DECLARE CONTOUR QC4 = &quot;SQUARE&quot;</td>
<td>Definition of the contour designator for the program &quot;SQUARE&quot;</td>
</tr>
<tr>
<td>8 QC10 = ( QC 1</td>
<td>QC 2 ) \ QC 3 \ QC 4</td>
</tr>
<tr>
<td>9 END PGM MODEL MM</td>
<td></td>
</tr>
</tbody>
</table>
### Contour description programs:

<table>
<thead>
<tr>
<th>Program Number</th>
<th>Contour Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 BEGIN PGM CIRCLE 1 MM</td>
<td>Contour description program: circle at right</td>
</tr>
<tr>
<td>1 CC X+65 Y+50</td>
<td></td>
</tr>
<tr>
<td>2 L PR+25 PA+0 R0</td>
<td></td>
</tr>
<tr>
<td>3 CP IPA+360 DR+</td>
<td></td>
</tr>
<tr>
<td>4 END PGM CIRCLE 1 MM</td>
<td></td>
</tr>
<tr>
<td>0 BEGIN PGM CIRCLE31XY MM</td>
<td>Contour description program: circle at left</td>
</tr>
<tr>
<td>1 CC X+Q1 Y+Q2</td>
<td></td>
</tr>
<tr>
<td>2 LP PR+Q3 PA+0 R0</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>
<tr>
<td>0 BEGIN PGM TRIANGLE MM</td>
<td>Contour description program: triangle at right</td>
</tr>
<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>
<tr>
<td>0 BEGIN PGM SQUARE MM</td>
<td>Contour description program: square at left</td>
</tr>
<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>
Fixed Cycles: Contour Pocket with Contour Formula

9.2 SL cycles with simple contour formula

Fundamentals
SL cycles and the simple contour formula enable you to form contours by combining up to 9 subcontours (pockets or islands) in a simple manner. You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the contour from the selected subcontours.

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

Program structure: Machining with SL cycles and complex contour formula

0 BEGIN PGM CONTDEF MM
...
5 CONTOUR DEF P1 = "POCK1.H" I2 = "ISLE2.H" DEPTH5 I3 = "ISLE3.H" DEPTH7.5
6 CYCL DEF 20 CONTOUR DATA...
8 CYCL DEF 22 ROUGH-OUT...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTDEF MM
Properties of the subcontours

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

Characteristics of the fixed cycles

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

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.
Fixed Cycles: Contour Pocket with Contour Formula

9.2  SL cycles with simple contour formula

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

- Show the soft-key row with special functions
- Select the menu for functions for contour and point machining
- Press the CONTOUR DEF soft key. The TNC opens the dialog for entering the contour formula
- Enter the name of the first subcontour. The first subcontour must always be the deepest pocket. Confirm with the ENT key
- Specify via soft key whether the next subcontour is a pocket or an island. Confirm with the ENT key
- Enter the name of the second subcontour. Confirm with the ENT key
- If needed, enter the depth of the second subcontour. Confirm with the ENT key
- Carry on with the dialog as described above until you have entered all subcontours.

Always start the list of subcontours with the deepest pocket!
If the contour is defined as an island, the TNC interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
If the depth is entered as 0, then for pockets the depth defined in the Cycle 20 is effective. Islands then rise up to the workpiece top surface!

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", page 197).
Cycles: Coordinate Transformations
10.1 Fundamentals

Overview

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The TNC provides the following coordinate transformation cycles:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>DATUM</td>
<td>263</td>
</tr>
<tr>
<td>247</td>
<td>DATUM SETTING</td>
<td>269</td>
</tr>
<tr>
<td>8</td>
<td>MIRRORING</td>
<td>270</td>
</tr>
<tr>
<td>10</td>
<td>ROTATION</td>
<td>272</td>
</tr>
<tr>
<td>11</td>
<td>SCALING FACTOR</td>
<td>274</td>
</tr>
<tr>
<td>26</td>
<td>AXIS-SPECIFIC SCALING</td>
<td>275</td>
</tr>
<tr>
<td>19</td>
<td>WORKING PLANE</td>
<td>277</td>
</tr>
</tbody>
</table>

For shifting contours directly within the program or from datum tables.

Datum setting during program run.

Mirroring contours.

Rotating contours in the working plane.

Increasing or reducing the size of contours.

Increasing or reducing the size of contours with axis-specific scaling.

Machining in tilted coordinate system on machines with swivel heads and/or rotary tables.

Effect of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM block (depending on machine parameter clearMode).
- Select a new program
10.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54)

**Effect**
A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.
When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

**Resetting**
- Program a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.

**Cycle parameters**
- **Datum shift**: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to six NC axes, each from –99999.9999 to 99999.9999

**NC blocks**

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>CYCL DEF 7.0 DATUM</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>CYCL DEF 7.1 X+60</td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>CYCL DEF 7.2 Y+40</td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>CYCL DEF 7.3 Z-5</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
10.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

Effect
Datum tables are used for:
- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.

Resetting
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
- Execute a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition

Status displays
In the additional status display, the following data from the datum table are shown:
- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number
Please note while programming:

**Danger of collision!**

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

If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program. If you work without **SEL TABLE**, then you must activate the desired datum table before the test run or the program run. (This applies also to the programming graphics).

- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table receives the status S.
- Use the file management in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes to select the desired table for program run: The table receives the status M.

The coordinate values from datum tables are only effective with absolute coordinate values.

New lines can only be inserted at the end of the table.

If you create datum tables, the file name has to start with a letter.

**Cycle parameters**

- **Datum shift**: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number entered in the Q parameter. Input range: 0 to 9999

**NC blocks**

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>77 CYCL DEF 7.0 DATUM SHIFT</td>
<td></td>
</tr>
<tr>
<td>78 CYCL DEF 7.1 #5</td>
<td></td>
</tr>
</tbody>
</table>
Cycles: Coordinate Transformations

10.3 DATUM SHIFT with datum tables (Cycle 7)

Selecting a datum table in the part program
With the SEL TABLE function you select the table from which the TNC takes the datums:

- Select the functions for program call: Press the PGM CALL key
- Press the DATUM TABLE soft key
- Select the complete path name of the datum table or the file with the SELECT soft key and confirm your entry with the END key

Program a SEL TABLE block before Cycle 7 Datum Shift.
A datum table selected with SEL TABLE remains active until you select another datum table with SEL TABLE or through PGM MGT.

Edit the datum table in the Programming mode of operation

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

Select the datum table in the Programming mode of operation

- Call the file manager: Press the PGM MGT key
- Display the datum tables: Press the SELECT TYPE and SHOW .D soft keys
- Select the desired table or enter a new file name.
- Edit the file The functions displayed in the soft-key row for editing include:
<table>
<thead>
<tr>
<th>Soft key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEGIN</td>
<td>Select beginning of table</td>
</tr>
<tr>
<td>END</td>
<td>Select end of table</td>
</tr>
<tr>
<td>PAGE↑</td>
<td>Go to the previous page</td>
</tr>
<tr>
<td>PAGE↓</td>
<td>Go to next page</td>
</tr>
<tr>
<td>INSERT LINE</td>
<td>Insert line (only possible at the end of table)</td>
</tr>
<tr>
<td>DELETE LINE</td>
<td>Delete line</td>
</tr>
<tr>
<td>FIND</td>
<td>Find</td>
</tr>
<tr>
<td>BEGIN LINE ←</td>
<td>Go to beginning of line</td>
</tr>
<tr>
<td>END LINE →</td>
<td>Go to end of line</td>
</tr>
<tr>
<td>COPY FIELD</td>
<td>Copy the current value</td>
</tr>
<tr>
<td>PASTE FIELD</td>
<td>Insert the copied value</td>
</tr>
<tr>
<td>APPEND N LINES</td>
<td>Add the entered number of lines (datums) to the end of the table</td>
</tr>
</tbody>
</table>
Configuring the datum table

If you do not wish to define a datum for an active axis, press the DEL key. Then the TNC clears the numerical value from the corresponding input field.

You can change the properties of tables. Enter the code number 555343 in the MOD menu. The TNC then offers the EDIT FORMAT soft key if a table is selected. When you press this soft key, the TNC opens a pop-up window where the properties are shown for each column of the selected table. Any changes made only affect the open table.

To exit a datum table

Select a different type of file in file management and choose the desired file.

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

Status displays

In the additional status display, the TNC shows the values of the active datum shift.
10.4 DATUM SETTING (Cycle 247, DIN/ISO: G247)

Effect
With the DATUM SETTING cycle you can activate as the new datum a preset defined in a preset table. After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display
In the status display the TNC shows the active preset number behind the datum symbol.

Please note before programming:
When activating a datum from the preset table, the TNC resets the datum shift, mirroring, rotation, scaling factor and axis-specific scaling factor. If you activate preset number 0 (line 0), then you activate the datum that you last set in the Manual Operation or El. Handwheel operating mode.
Cycle 247 is not functional in Test Run mode.

Cycle parameters
- **Number for datum?** Enter the number of the desired datum from the preset table. Optionally you can use the soft key SELECT and select the desired datum from the preset table. Input range 0 to 65535

NC blocks

<table>
<thead>
<tr>
<th>13 CYCL DEF 247 DATUM SETTING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q339=4 ;DATUM NUMBER</td>
</tr>
</tbody>
</table>

Status displays
In the additional status display (POS. DISP. STATUS) the TNC shows the active preset number behind the datum dialog.
10.5 MIRRORING (Cycle 8, DIN/ISO: G28)

Effect
The TNC can machine the mirror image of a contour in the working plane.
The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in SL cycles).
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also “jumps” to another location.

Resetting
Program the MIRROR IMAGE cycle once again with NO ENT.
Please note while programming:

If you work in a tilted system with Cycle 8 the following procedure is recommended:
- First program the tilting movement and then call Cycle 8 MIRRORING!

Cycle parameters

- **Mirrored axis?**: Enter the axis to be mirrored. You can mirror all axes except for the spindle axis—including rotary axes—with the exception of the spindle axis and its associated auxiliary axis. You can enter up to three axes. Input range: Up to three NC axes X, Y, Z, U, V, W, A, B, C

NC blocks

| 79 CYCL DEF 8.0 MIRROR IMAGE |
| 80 CYCL DEF 8.1 X Y Z |
10.6 ROTATION (Cycle 10, DIN/ISO: G73)

Effect
The TNC can rotate the coordinate system about the active datum in the working plane within a program. The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

Reference axis for the rotation angle:
- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis

Resetting
Program the ROTATION cycle once again with a rotation angle of 0°.
Please note while programming:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary. After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

Cycle parameters

- **Rotation**: Enter the rotation angle in degrees (°). Input range $-360.000°$ to $+360.000°$ (absolute or incremental)

NC blocks

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>12 CALL LBL 1</td>
<td></td>
</tr>
<tr>
<td>13 CYCL DEF 7.0 DATUM SHIFT</td>
<td></td>
</tr>
<tr>
<td>14 CYCL DEF 7.1 X+60</td>
<td></td>
</tr>
<tr>
<td>15 CYCL DEF 7.2 Y+40</td>
<td></td>
</tr>
<tr>
<td>16 CYCL DEF 10.0 ROTATION</td>
<td></td>
</tr>
<tr>
<td>17 CYCL DEF 10.1 ROT+35</td>
<td></td>
</tr>
<tr>
<td>18 CALL LBL 1</td>
<td></td>
</tr>
</tbody>
</table>
10.7 SCALING (Cycle 11, DIN/ISO: G72)

Effect
The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on:
- all three coordinate axes at the same time
- Dimensions in cycles

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

Enlargement: SCL greater than 1 (up to 99.999 999)
Reduction: SCL less than 1 (down to 0.000 001)

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

Cycle parameters
- **Scaling factor**: Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under 'Effect' above). Input range 0.000001 to 99.999999

NC blocks

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>CALL LBL 1</td>
</tr>
<tr>
<td>12</td>
<td>CYCL DEF 7.0 DATUM</td>
</tr>
<tr>
<td>13</td>
<td>CYCL DEF 7.1 X+60</td>
</tr>
<tr>
<td>14</td>
<td>CYCL DEF 7.2 Y+40</td>
</tr>
<tr>
<td>15</td>
<td>CYCL DEF 11.0 SCALING</td>
</tr>
<tr>
<td>16</td>
<td>CYCL DEF 11.1 SCL 0.75</td>
</tr>
<tr>
<td>17</td>
<td>CALL LBL 1</td>
</tr>
</tbody>
</table>
10.8 AXIS-SPECIFIC SCALING (Cycle 26)

Effect
With Cycle 26 you can account for shrinkage and oversize factors for each axis.
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

Resetting
Program the SCALING cycle once again with a scaling factor of 1 for the same axis.

Please note while programming:

Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.
You can program each coordinate axis with its own axis-specific scaling factor.
In addition, you can enter the coordinates of a center for all scaling factors.
The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.
**Cycle parameters**

- **Axis and scaling factor**: Select the coordinate axis/axes by soft key and enter the factor(s) involved in enlarging or reducing. Input range 0.000001 to 99.999999
- **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range -99999.9999 to 99999.9999

**NC blocks**

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>CALL LBL 1</td>
</tr>
<tr>
<td>26</td>
<td>CYCL DEF 26.0 AXIS-SPECIFIC SCALING</td>
</tr>
<tr>
<td>27</td>
<td>CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20</td>
</tr>
<tr>
<td>28</td>
<td>CALL LBL 1</td>
</tr>
</tbody>
</table>
Effect

In Cycle 19 you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to 3 rotations (spatial angle) of the fixed machine coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.

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

If you program the position of the working plane via spatial angles, the TNC will calculate the required angle positions of the tilted axes automatically and will store these in the parameters Q120 (A axis) to Q122 (C axis). If two solutions are possible, the TNC will choose the shorter path from the zero position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You must move all axes to activate compensation for all axes.

If you set the function **Tilting program run** to **Active** in the Manual Operation mode, the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.
Please note while programming:

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the rotary axes or as angular components of a tilted plane. Refer to your machine manual.

Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

The working plane is always tilted around the active datum. If you use Cycle 19 when M120 is active, the TNC automatically rescinds the radius compensation, which also rescinds the M120 function.

Cycle parameters

- **Rotary axis and tilt angle?:** Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range -360.000 to 360.000

If the TNC automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate? F?:** Traverse speed of the rotary axis during automatic positioning. Input range: 0 to 99999.999

- **Set-up clearance? (incremental value):** The TNC positions the tilting head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range: 0 to 99999.999
Resetting
To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle once again and respond to the dialog question with the NO ENT key to disable the function.

Positioning the axes of rotation

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

Manual positioning of rotary axes
If the rotary axes are not positioned automatically in Cycle 19, you must position them in a separate L block after the cycle definition. If you use axis angles, you can define the axis values right in the L block. If you use spatial angles, then use the Q parameters Q120 (A-axis value), Q121 (B-axis value) and Q122 (C-axis value), which are described by Cycle 19.

For manual positioning, always use the rotary axis positions stored in Q parameters Q120 to Q122. Avoid using functions, such as M94 (modulo rotary axes), in order to avoid discrepancies between the actual and nominal positions of rotary axes in multiple definitions.

Example NC blocks:

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</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>
<tr>
<td>13</td>
<td>CYCL DEF 19.1 A+0 B+45 C+0</td>
</tr>
<tr>
<td>14</td>
<td>L A+Q120 C+Q122 R0 F1000</td>
</tr>
<tr>
<td>15</td>
<td>L Z+80 R0 FMAX</td>
</tr>
<tr>
<td>16</td>
<td>L X-8.5 Y-10 R0 FMAX</td>
</tr>
</tbody>
</table>
Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle 19:

- The TNC can position only controlled axes.
- In order for the tilted axes to be positioned, you must enter a feed rate and a set-up clearance in addition to the tilting angles, during cycle definition.
- Use only preset tools (the full tool length must be defined).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting.
- The TNC performs the tilt at the last programmed feed rate.
  The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

Example NC blocks:

```
10 L Z+100 R0 FMAX
11 L X+25 Y+10 R0 FMAX
12 CYCL DEF 19.0 WORKING PLANE
13 CYCL DEF 19.1 A+0 B+45 C+0 F5000 SETUP50
14 L Z+80 R0 FMAX
15 L X-8.5 Y-10 R0 FMAX
```

Position display in the tilted system

On activation of Cycle 19, the displayed positions (ACTL and NOML) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If necessary, the TNC outputs an error message.
Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system.

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92) can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

1. Activate the datum shift
2. Activate tilting function
3. Activate rotation
...
Workpiece machining
...
1. Reset the rotation
2. Reset the tilting function
3. Reset the datum shift
10 Cycles: Coordinate Transformations

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

Procedure for working with Cycle 19 WORKING PLANE

1 Write the program
- Define the tool (not required if TOOL.T is active), and enter the full tool length.
- Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece or clamping devices during tilting.
- If required, position the rotary axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes.
- Traverse all principal axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- If required, define Cycle 19 WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle 19. You can define the new angular values directly.
- Reset Cycle 19 WORKING PLANE; program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle 19 and answer the dialog question with NO ENT.
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

2 Clamp the workpiece

3 Datum setting
- Manually by touch-off
- Controlled with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles User’s Manual, chapter 2).
- Automatically with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles User’s Manual, chapter 3).

4 Start the part program in the operating mode Program Run, Full Sequence

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

Example: Coordinate transformation cycles

Program sequence
- Program the coordinate transformations in the main program
- Machining within a subprogram

Program sequence:

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM COTRANS MM</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>BLK FORM 0.1 Z X+0 Y+0 Z-20</td>
<td>Definition of workpiece blank</td>
</tr>
<tr>
<td>2</td>
<td>BLK FORM 0.2 X+130 Y+130 Z+0</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>TOOL CALL 1 Z S4500</td>
<td>Tool call</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 7.0 DATUM SHIFT</td>
<td>Shift datum to center</td>
</tr>
<tr>
<td>6</td>
<td>CYCL DEF 7.1 X+65</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>CYCL DEF 7.2 Y+65</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>CALL LBL 1</td>
<td>Call milling operation</td>
</tr>
<tr>
<td>9</td>
<td>LBL 10</td>
<td>Set label for program section repeat</td>
</tr>
<tr>
<td>10</td>
<td>CYCL DEF 10.0 ROTATION</td>
<td>Rotate by 45° (incremental)</td>
</tr>
<tr>
<td>11</td>
<td>CYCL DEF 10.1 IROT+45</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>CALL LBL 1</td>
<td>Call milling operation</td>
</tr>
<tr>
<td>13</td>
<td>CALL LBL 10 REP 6/6</td>
<td>Return jump to LBL 10; repeat the milling operation six times</td>
</tr>
<tr>
<td>14</td>
<td>CYCL DEF 10.0 ROTATION</td>
<td>Reset the rotation</td>
</tr>
<tr>
<td>15</td>
<td>CYCL DEF 10.1 ROT+0</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>CYCL DEF 7.0 DATUM SHIFT</td>
<td>Reset the datum shift</td>
</tr>
<tr>
<td>17</td>
<td>CYCL DEF 7.1 X+0</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>CYCL DEF 7.2 Y+0</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>L Z+250 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>20</td>
<td>LBL 1</td>
<td>Subprogram 1</td>
</tr>
<tr>
<td>21</td>
<td>L X+0 Y+0 R0 FMAX</td>
<td>Define milling operation</td>
</tr>
<tr>
<td>22</td>
<td>L Z+2 R0 FMAX M3</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>L Z-5 R0 F200</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>L X+30 RL</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>L IY+10</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>RND R5</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>L IX+20</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>L IX+10 IY-10</td>
<td></td>
</tr>
</tbody>
</table>
### Cycles: Coordinate Transformations

#### 10.10 Programming Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
<td>RND R5</td>
</tr>
<tr>
<td>30</td>
<td>L IX-10 IY-10</td>
</tr>
<tr>
<td>31</td>
<td>L IX-20</td>
</tr>
<tr>
<td>32</td>
<td>L IY+10</td>
</tr>
<tr>
<td>33</td>
<td>L X+0 Y+0 R0 F5000</td>
</tr>
<tr>
<td>34</td>
<td>L Z+20 R0 FMAX</td>
</tr>
<tr>
<td>35</td>
<td>LBL 0</td>
</tr>
<tr>
<td>36</td>
<td>END PGM COTRANS MM</td>
</tr>
</tbody>
</table>
11

Cycles: Special Functions
11 Cycles: Special Functions

11.1 Fundamentals

Overview

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

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>9 DWELL TIME</td>
<td>287</td>
</tr>
<tr>
<td>12</td>
<td>12 PROGRAM CALL</td>
<td>288</td>
</tr>
<tr>
<td>12</td>
<td>13 SPINDLE ORIENTATION</td>
<td>290</td>
</tr>
<tr>
<td>32</td>
<td>32 TOLERANCE</td>
<td>291</td>
</tr>
<tr>
<td>225</td>
<td>225 ENGRAVING of texts</td>
<td>309</td>
</tr>
<tr>
<td>291</td>
<td>291 COUPLING TURNING INTERPOLATION</td>
<td>303</td>
</tr>
<tr>
<td>292</td>
<td>292 CONTOUR TURNING INTERPOLATION</td>
<td>294</td>
</tr>
<tr>
<td>232</td>
<td>232 FACE MILLING</td>
<td>313</td>
</tr>
<tr>
<td>239</td>
<td>239 ASCERTAIN THE LOAD</td>
<td>318</td>
</tr>
</tbody>
</table>
11.2 DWELL TIME (Cycle 9, DIN/ISO: G04)

Function
This causes the execution of the next block within a running program to be delayed by the programmed DWELL TIME. A dwell time can be used for such purposes as chip breaking. The cycle becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.

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

NC blocks

| 89 CYCL DEF 9.0 DWELL TIME |
| 90 CYCL DEF 9.1 DWELL 1.5 |
11.3 PROGRAM CALL (Cycle 12, DIN/ISO: G39)

Cycle function
Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs. These can then be called like fixed cycles.

Please note while programming:

- The program you are calling must be stored in the internal memory of your TNC.
- If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only enter the program name.
- If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example TNC:\KLAR35\FK1\50.H.
- If you want to define a DIN/ISO program to be a cycle, enter the file type .I behind the program name.
- As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called program can also influence the calling program.
Cycle parameters

- **Program name**: Enter the name of the program you want to call and, if necessary, the directory it is located in or
- Activate the file select dialog with the SELECT soft key and select the program to be called

Call the program with:
- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Designate program 50 as a cycle and call it with M99

| 55 CYCL DEF 12.0 PGM CALL |
| 56 CYCL DEF 12.1 PGM TNC: "KLAR35\FK1\50.H" |
| 57 L X+20 Y+50 FMAX M99 |
Cycle function

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

The TNC can control the machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for:
- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle at an angle that has been set by the machine tool builder.


Please note while programming:

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

Cycle parameters

- **Angle of orientation**: Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°
11.5  TOLERANCE (Cycle 32, DIN/ISO: G62)

Cycle function

With the entries in Cycle 32 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the TNC has been adapted to the machine’s characteristics.

The TNC automatically smoothens the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without short pauses for computing time. **Even if the TNC does not move with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the TNC can move the axes.

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

Influences of the geometry definition in the CAM system

The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The maximum point spacing of NC programs generated in a postprocessor (PP) is defined through the chord error. If the chord error is less than or equal to the tolerance value T defined in Cycle 32, then the TNC can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing if in Cycle 32 you choose a tolerance value between 110-% and 200-% of the CAM chord error.
Please note while programming:

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the TNC, but by the fact that, in order to machine the contour element transitions very exactly, the TNC might have to drastically reduce the speed.

Cycle 32 is DEF active which means that it becomes effective as soon as it is defined in the part program.

The TNC resets Cycle 32 if you
- Redefine it and confirm the dialog question for the tolerance value with NO ENT.
- Select a new program with the PGM MGT key.

After you have reset Cycle 32, the TNC reactivates the tolerance that was predefined by machine parameter.

In a program with millimeters set as unit of measure, the TNC interprets the entered tolerance value in millimeters. In an inch program it interprets it as inches.

If you transfer a program with Cycle 32 that contains only the cycle parameter Tolerance value T, the TNC inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases, except if HSC filters are active on your machine (settings made by the machine tool builder).

If Cycle 32 is active, the TNC shows the parameters defined for Cycle 32 on the CYC tab of the additional status display.
Cycle parameters

- **Tolerance value T**: Permissible contour deviation in mm (or inches with inch programming). Input range 0 to 99999.9999
- **HSC MODE, Finishing=0, Roughing=1**: Activate filter:
  - Input value 0: **Milling with increased contour accuracy**. The TNC uses internally defined finishing filter settings
  - Input value 1: **Milling at an increased feed rate**. The TNC uses internally defined roughing filter settings
- **Tolerance for rotary axes TA**: Permissible position error of rotary axes in degrees when M128 is active (FUNCTION TCPM). The TNC always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for programs for more than one axis by entering a large tolerance value (e.g., 10°), since the TNC does not always have to move the rotary axis to the given nominal position. The contour will not be damaged by entering a rotary axis tolerance value. Only the position of the rotary axis with respect to the workpiece surface will change. Input range 0 to 179.9999

**NC blocks**

<table>
<thead>
<tr>
<th>NC block</th>
</tr>
</thead>
<tbody>
<tr>
<td>95 CYCL DEF 32.0 TOLERANCE</td>
</tr>
<tr>
<td>96 CYCL DEF 32.1 T0.05</td>
</tr>
<tr>
<td>97 CYCL DEF 32.2 HSC-MODE:1 TA5</td>
</tr>
</tbody>
</table>
11.6 CONTOUR TURNING INTERPOLATION (Cycle 292, DIN/ISO: G292, software option 96)

Cycle run
Cycle 292 CONTOUR TURNING INTERPOLATION couples the tool spindle to the position of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. Cycle 292 CONTOUR TURNING INTERPOLATION is run in milling mode and is CALL-active. After executing this cycle, the TNC deactivates the spindle coupling again.

Before using Cycle 292, you first need to define the desired contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. By entering Q560=1 you can turn the contour; a cutting edge of the tool is oriented to the center of a circle. When you enter Q560=0 you can mill the contour; the spindle is not oriented in this case.

**Cycle run, Q560=1: Contour turning**

1. The TNC first stops the spindle (M5).
2. The TNC orients the tool spindle to the specified center of rotation. The specified angle Q336 is taken into account. If defined, the "ORI" value from the turning tool table (toolturn.trn) is also considered.
3. The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4. The TNC positions the tool to the contour start radius Q491, taking the selected machining operation inside/outside Q529 and the clearance to side Q357 into account. The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**
5. The TNC uses interpolation turning to machine the defined contour. In interpolation turning the linear axes of the working plane move on a circle, whereas the spindle axis is oriented perpendicularly to the surface.
6. At the end point of the contour, the TNC retracts the tool perpendicularly by the set-up clearance.
7. Finally, the TNC retracts the tool to the clearance height.
8. The TNC now automatically deactivates the coupling of the tool spindle to the linear axes.
Cycle run, Q560=0: Contour milling

1. The M3/M4 function programmed before the cycle call remains in effect.
2. No spindle stop and no spindle orientation are performed. Q336 is not taken into account.
3. The TNC positions the tool to the contour start radius Q491, taking the selected machining operation inside/outside Q529 and the clearance to side Q357 into account. The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**
4. The TNC machines the defined contour with the spindle rotating (M3/M4). In the process, the reference axes of the working plane move on a circle; the TNC does not orient the tool spindle.
5. At the end point of the contour, the TNC retracts the tool perpendicularly by the set-up clearance.
6. Finally, the TNC retracts the tool to the clearance height.
Cycles: Special Functions

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

Please note while programming:
A programming example is provided at the end of this chapter, see page 322.

Program the contour either with monotonically decreasing or monotonically increasing coordinates. When programming, remember to use only positive radius values.

Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

Roughing operations with multiple passes are not possible in this cycle.

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

For inside machining, the TNC checks whether the active tool radius is smaller than half the contour start diameter Q491 plus the clearance to side Q357. If the check shows that the tool is too large, the program is aborted.

If Cycle 8 MIRRORING is active, the TNC does not execute the interpolation turning cycle.

If Cycle 26 SCALING FACTOR is active and the scaling factor in an axis is not equal to 1, the TNC does not execute the interpolation turning cycle.
The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

The center of the turning contour is the starting point in the working plane at the time the cycle is called.

This cycle is effective only for machines with servo-controlled spindle. Software option 96 must be enabled. If Q560=1 the TNC does not check whether the spindle is rotating when the cycle is run. (Independent of CfgGeoCycle – displaySpindleError)

Your TNC might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.
Cycles: Special Functions

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

Cycle parameters

- **Spindle coupling (0, 1)** Q560: Define whether the spindle is to be coupled.
  - 0: Spindle coupling off (contour milling)
  - 1: Spindle coupling on (contour turning)

- **Angle of spindle** Q336: The TNC orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool and defined the value "ORI" in the turning tool (toolturn.trn), then this value will also be accounted for when orienting the spindle. Input range 0.000 to 360.000

- **Change tool direction (3, 4)** Q546: Direction of spindle rotation of the active tool:
  - 3: Right-turning tool (M3)
  - 4: Left-turning tool (M4)

- **Machining operation (+1, 0)** Q529: Define whether to perform inside or outside machining:
  - +1: Inside machining
  - 0: Outside machining

- **Surface oversize** Q221: Allowance in the working plane. Input range 0 to 99.9999

- **Infeed** Q441 (mm/rev): Amount that the tool advances with each revolution. Input range 0.001 to 99.999

- **Feed rate** Q449 (mm/min): Feed rate with respect to the contour starting point Q491. Input range 0.1 to 99999.9. The feed rate of the tool’s center point path is adjusted according to the tool radius and the machining operation Q529. From these parameters, the TNC determines the programmed cutting speed at the diameter of the contour starting point.
  - Q529=1: The feed rate of the tool center point path is reduced for inside machining
  - Q529=0: The feed rate of the tool center point path is increased for outside machining

- **Contour start radius** Q491 (absolute value): Radius of the contour starting point (e.g. X coordinate, with tool axis Z). Input range 0.9999 to 999999.9999

- **Clearance to side** Q357 (incremental): Clearance between the side of the tool and the workpiece when approaching for the first plunging depth. Input range 0 to 999999.9

- **Clearance height** Q445 (absolute): Absolute height at which the tool cannot collide with the workpiece. The tool retracts to this position at the end of the cycle. Input range -999999.9999 to 999999.9999

NC blocks

<table>
<thead>
<tr>
<th>63 CYCL DEF 292 CONTOUR TURNING INTERPOLATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q560=1 ;SPINDLE COUPLING</td>
</tr>
<tr>
<td>Q336=0 ;ANGLE OF SPINDLE</td>
</tr>
<tr>
<td>Q546=3 ;CHANGE TOOL DIRECTN.</td>
</tr>
<tr>
<td>Q529=0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q221=0 ;SURFACE OVERSIZE</td>
</tr>
<tr>
<td>Q441=0.5 ;INFEED</td>
</tr>
<tr>
<td>Q449=2000;FEED RATE</td>
</tr>
<tr>
<td>Q491=0 ;CONTOUR START DIA.</td>
</tr>
<tr>
<td>Q357=2 ;CLEARANCE TO SIDE</td>
</tr>
<tr>
<td>Q445=50 ;CLEARANCE HEIGHT</td>
</tr>
</tbody>
</table>
Machining variants

Before using Cycle 292, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

<table>
<thead>
<tr>
<th>Tool axis used</th>
<th>Axial coordinate</th>
<th>Radial coordinate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>Z</td>
<td>X</td>
</tr>
<tr>
<td>X</td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Z</td>
</tr>
</tbody>
</table>

**Example:** If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius of the contour in X.

You can use this cycle for inside and outside machining. The following information illustrates some of the notes given in the "Please note while programming" section. You can also find a programming example in "Example: Interpolation Turning Cycle 292", page 322

**Inside machining**

- The center of rotation is the position of the tool in the working plane at the time the cycle is called 1

- **After the cycle is started, neither the indexable insert nor the spindle center must be moved into the center of rotation!** Keep this in mind when describing the contour! 2

- The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

When programming an inside contour, please also remember:

- Program either monotonically increasing radial and axial coordinates, e.g. 1-5
- Or program monotonically decreasing radial and axial coordinates, e.g. 5-1
- Program inside contours with a radius greater than the tool radius.
Outside machining

- The center of rotation is the position of the tool in the working plane at the time the cycle is called.
- After the cycle is started, neither the indexable insert nor the spindle center must be moved into the center of rotation. Keep this in mind when describing the contour!
- The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the TNC positions the tool at rapid traverse in the tool axis direction to the contour starting point! Make sure that there is no material at the contour starting point!

When programming an outside contour, please also remember:
- Program either monotonically increasing radial and monotonically decreasing axial coordinates, e.g. 1-5
- Or program monotonically decreasing radial and monotonically increasing axial coordinates, e.g. 5-1
- Program outside contours with a radius greater than 0.
Defining the tool

Overview

Depending on the setting of the parameter Q560, you can mill (Q560=0) or turn (Q560=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

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

These three possibilities of defining the tool are described in more detail below:

- Define a turning tool in the tool table (tool.t) as a milling tool

If you are working without option 50, define the turning tool in the tool table (tool.t) as a milling cutter. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). Orient the turning tool to the spindle center and enter this spindle orientation angle in the parameter Q336 of the cycle. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

The tool holder is not monitored! If the rotation diameter resulting from the tool holder is greater than that from the cutting edge, the machine operator must take this into account for inside machining.

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

You can use a milling cutter for interpolation turning. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). Orient a cutting edge of the milling cutter to the spindle center and enter this angle in the parameter Q336. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

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

If you are working with option 50, you can define the turning tool in the turning tool table (toolturn.trn). In this case, the spindle is oriented to the center of rotation by taking tool-specific data into account, such as the machining operation (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and the parameter Q336.
Cycles: Special Functions

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

The spindle orientation is calculated as follows:

<table>
<thead>
<tr>
<th>Machining</th>
<th>TO</th>
<th>Spindle orientation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interpolation turning, outside</td>
<td>1</td>
<td>ORI + Q336</td>
</tr>
<tr>
<td>Interpolation turning, inside</td>
<td>7</td>
<td>ORI + Q336 + 180</td>
</tr>
<tr>
<td>Interpolation turning, outside</td>
<td>7</td>
<td>ORI + Q336 + 180</td>
</tr>
<tr>
<td>Interpolation turning, inside</td>
<td>1</td>
<td>ORI + Q336</td>
</tr>
<tr>
<td>Interpolation turning, outside</td>
<td>8,9</td>
<td>ORI + Q336</td>
</tr>
<tr>
<td>Interpolation turning, inside</td>
<td>8,9</td>
<td>ORI + Q336</td>
</tr>
</tbody>
</table>

You can use the following tool types for interpolation turning:
- TYPE: ROUGH, with the machining directions TO: 1 or 7
- TYPE: FINISH, with the machining directions TO: 1 or 7
- TYPE: BUTTON, with the machining directions TO: 1 or 7

For inside machining, the TNC checks whether the active tool radius is smaller than half the contour start diameter Q491 plus the clearance to side Q357. If the check shows that the tool is too large, the program is aborted.

The following tool types cannot be used for interpolation turning: (error message "Function not possible with this tool type" is displayed)
- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD
11.7 COUPLING INTERPOLATION TURNING (cycle 291, DIN/ISO: G291, software option 96)

Cycle run

Cycle 291 COUPLING TURNING INTERPOLATION couples the tool spindle to the position of the linear axes or deactivates this spindle coupling. In interpolation turning the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates Q216 and Q217. Cycle 291 COUPLING TURNING INTERPOLATION is run in milling mode and is CALL-active.

Cycle run if Q560=1:
1 The TNC first stops the spindle (M5).
2 The TNC orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation Q336 is taken into account. If an "ORI" value is given in the tool table, it is also taken into account if so defined.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 To terminate the cycle, the coupling must be deactivated by the operator. (With Cycle 291 and end of program/internal stop resp.)

Cycle run if Q560=0:
1 The TNC deactivates the spindle coupling.
2 The tool spindle is no longer coupled to the position of the linear axes.
3 Machining with the interpolation turning cycle 291 is terminated.
4 If Q560=0, the parameters Q336, Q216, Q217 are irrelevant.

Please note while programming:

After defining Cycle 291 and CYCLE CALL program the operation you wish to perform. In order to describe the circular motions of the linear axes linear/polar coordinates are used, for example. A programming example is provided at the end of this section, see page 320.
Cycle 291 is CALL-active. Programming of M3/M4 is not required. In order to describe the circular motions of the linear axes, you can use \texttt{CC} and \texttt{C} coordinates, for example.

If you specify the turning tool in the turning tool table (toolturn.trn), we recommend to work with parameter Q561=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort.

For programming Q561=1 you can work with a radius compensation \texttt{RR} or \texttt{RL}. (However, if you are programming parameter Q561=0, you will have to do without a radius compensation \texttt{RR} or \texttt{RL} for the description of your contour. In addition, while programming you have to make sure that the motion of the center of the tool TCP is programmed without spindle coupling. This kind of programming is much more complex!)

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

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

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

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

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

If Cycle 8 MIRRORING is active, the TNC does not execute the interpolation turning cycle.

If Cycle 26 SCALING FACTOR is active and the scaling factor in an axis is not equal to 1, the TNC does not execute the interpolation turning cycle.

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

Your TNC might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

Software option 96 must be enabled.
Cycle parameters

- **Spindle coupling (0, 1)** Q560: Define whether the tool spindle is coupled to the position of the linear axes. When spindle coupling is active, a cutting edge of the tool is oriented to the center of rotation.
  - 0: Spindle coupling off
  - 1: Spindle coupling on

- **Angle of spindle** Q336: The TNC orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool and defined the value "ORI" in the turning tool (toolturn.trn), then this value will also be accounted for when orienting the spindle. Input range 0.000 to 360.000

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

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

- **Convert turning tool** Q561 (0/1): Only relevant if you define your tool in the turning tool table (toolturn.trn). With this parameter you decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool.
  - 0: No change—the turning tool is interpreted as described in the turning tool table (toolturn.trn). In this case you may not use radius compensation RR or RL. You also have to make sure during programming that the motion of the center of the tool TCP is described without spindle coupling. This type of programming is much more complex.
  - 1: The value XL of the turning tool table (toolturn.trn) is interpreted like a radius R of a milling tool table. This way you are able to use radius compensation RR or RL to program your contour. This kind of programming is recommended.

### NC blocks

<table>
<thead>
<tr>
<th>64 CYL DEF 291 COUPLING TURNING INTERPOLATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q560=1 ;SPINDLE COUPLING</td>
</tr>
<tr>
<td>Q336=0 ;ANGLE OF SPINDLE</td>
</tr>
<tr>
<td>Q216=50 ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q217=50 ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q561=1 ;TURNING TOOL CONVERSION</td>
</tr>
</tbody>
</table>
Defining the tool

Overview

Depending on the setting of the parameter Q560, you can activate (Q560=1) or deactivate (Q560=0) the COUPLING TURNING INTERPOLATION cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter Q560 to 1, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

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

These three possibilities of defining the tool are described in more detail below:

- Define a turning tool in the tool table (tool.t) as a milling tool

If you are working without option 50, define the turning tool in the tool table (tool.t) as a milling cutter. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Orient the turning tool to the spindle center and enter this spindle orientation angle in the parameter Q336 of the cycle. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.

The tool holder is not monitored! If the rotation diameter resulting from the tool holder is greater than that from the cutting edge, the machine operator must take this into account for inside machining.

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

You can use a milling cutter for interpolation turning. In this case, the following data from the tool table are taken into account (including delta values): Length (L), radius (R) and toroid cutter radius (R2). Orient a cutting edge of the milling cutter to the spindle center and enter this angle in the parameter Q336. For outside machining, the spindle orientation Q336 is used; for inside machining, the spindle orientation is calculated from Q336+180.
Define a turning tool in the turning tool table (toolturn.trn)

If you are working with option 50, you can define the turning tool in the turning tool table (toolturn.trn). In this case, the spindle is oriented to the center of rotation by taking tool-specific data into account, such as the machining operation (TO in the turning tool table), the orientation angle (ORI in the turning tool table), and the parameters Q336 and Q561.

If you specify the turning tool in the turning tool table (toolturn.trn), we recommend to work with parameter Q561=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. For programming Q561=1 you can work with a radius compensation RR or RL. (However, if you are programming parameter Q561=0, you will have to do without a radius compensation RR or RL for the description of your contour. In addition, while programming you have to make sure that the motion of the center of the tool TCP is programmed without spindle coupling. This kind of programming is much more complex!)

If you programmed parameter Q561=1, you must program the following in order to complete the operation of interpolation turning:
- R0, overrides radius compensation
- Cycle 291 with parameters Q560=0 and Q561=0, overrides spindle coupling
- CYCLE CALL, for calling up Cycle 291
- TOOL CALL overrides the conversion of parameter Q561

If you programmed parameter Q561=1, you may only use the following types of tools:
- TYPE: ROUGH, FINISH, BUTTON with the machining directions TO: 1 or 8, XL>=0
- TYPE: ROUGH, FINISH, BUTTON with the machining directions TO: 7, XL<=0

The spindle orientation is calculated as follows:

<table>
<thead>
<tr>
<th>Machining</th>
<th>TO</th>
<th>Spindle orientation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interpolation turning, outside</td>
<td>1</td>
<td>ORI + Q336</td>
</tr>
<tr>
<td>Interpolation turning, inside</td>
<td>7</td>
<td>ORI + Q336 + 180</td>
</tr>
<tr>
<td>Interpolation turning, outside</td>
<td>7</td>
<td>ORI + Q336 + 180</td>
</tr>
<tr>
<td>Interpolation turning, inside</td>
<td>1</td>
<td>ORI + Q336</td>
</tr>
<tr>
<td>Interpolation turning, outside</td>
<td>8</td>
<td>ORI + Q336</td>
</tr>
<tr>
<td>Interpolation turning, inside</td>
<td>8</td>
<td>ORI + Q336</td>
</tr>
</tbody>
</table>
Cycles: Special Functions

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

You can use the following tool types for interpolation turning:
- TYPE: ROUGH, with the machining directions TO: 1, 7, 8
- TYPE: FINISH, with the machining directions TO: 1, 7, 8
- TYPE: BUTTON, with the machining directions TO: 1, 7, 8

The following tool types cannot be used for interpolation turning: (error message "Function not possible with this tool type" is displayed)
- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD
11.8 ENGRAVING (Cycle 225, DIN/ISO: G225)

**Cycle run**
This cycle is used to engrave texts on a flat surface of the workpiece. The texts can be arranged in a straight line or along an arc.

1. The TNC positions the tool in the working plane to the starting point of the first character.
2. The tool plunges perpendicularly to the engraving floor and mills the character. The TNC retracts the tool to the set-up clearance between the characters when required. After machining the character, the tool is at the set-up clearance above the workpiece surface.
3. This process is repeated for all characters to be engraved.
4. Finally, the TNC retracts the tool to the 2nd set-up clearance.

**Please note while programming:**

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

If you engrave the text in a straight line (Q516=0), the starting point of the first character is determined by the tool position at the time the cycle is called.

If you engrave the text along an arc (Q516=1), the arc’s center is determined by the tool position at the time the cycle is called.

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

- **Engraving text** QS500: Text to be engraved inside quotation marks. Assignment of a string variable through the Q key of the numerical keypad. The Q key on the ASCI keyboard represents normal text input. Allowed entry characters: see "Engraving system variables", page 312

- **Character height** Q513 (absolute): Height of the characters to be engraved in mm. Input range 0 to 99999.9999

- **Space factor** Q514: The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the TNC if you program Q514 = 0. If Q514 is not equal to 0, the TNC scales the space between the characters. Input range 0 to 9.9999

- **Font** Q515: Currently without function

- **Text in a straight line/on arc (0/1)** Q516: Engrave text in a straight line: Input = 0 Engrave text on an arc: Input = 1

- **Angle of rotation** Q374: Center angle if the text is to be arranged on an arc. Engraving angle if the text is arranged in a straight line. Input range -360.0000 to +360.0000°

- **Radius of text on an arc** Q517 (absolute): Radius of the arc in mm on which the TNC is to arrange the text. Input range 0 to 99999.9999

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

- **Depth** Q201 (incremental value): Distance between workpiece surface and engraving floor

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

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

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

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

---

### NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>62 CYCL DEF 225 ENGRAVING</td>
</tr>
<tr>
<td>Q500=&quot;A&quot; ;ENGRAVING TEXT</td>
</tr>
<tr>
<td>Q513=10 ;CHARACTER HEIGHT</td>
</tr>
<tr>
<td>Q514=0 ;SPACE FACTOR</td>
</tr>
<tr>
<td>Q515=0 ;FONT</td>
</tr>
<tr>
<td>Q516=0 ;TEXT LAYOUT</td>
</tr>
<tr>
<td>Q374=0 ;ANGLE OF ROTATION</td>
</tr>
<tr>
<td>Q517=0 ;CIRCLE RADIUS</td>
</tr>
<tr>
<td>Q207=750 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q201=-0.5 ;DEPTH</td>
</tr>
<tr>
<td>Q206=150 ;FEED RATE FOR PLUNGING</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q203=+20 ;SURFACE COORDINATE</td>
</tr>
<tr>
<td>Q204=50 ;2ND SET-UP CLEARANCE</td>
</tr>
</tbody>
</table>
Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers:

! # $ % & ' ( ) * + - . / : ; < = > ? @ [ \ ] _ ß CE

The TNC uses the special characters % and \ for special functions. These characters must be indicated twice in the text to be engraved (e.g. %%) if you want to engrave them.

When engraving German umlauts, ß, ø, @ or the CE character, enter the character % before the character to be engraved:

<table>
<thead>
<tr>
<th>Algebraic sign</th>
<th>Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>ä</td>
<td>%ae</td>
</tr>
<tr>
<td>ö</td>
<td>%oe</td>
</tr>
<tr>
<td>ü</td>
<td>%ue</td>
</tr>
<tr>
<td>Ä</td>
<td>%AE</td>
</tr>
<tr>
<td>Ö</td>
<td>%OE</td>
</tr>
<tr>
<td>Ù</td>
<td>%UE</td>
</tr>
<tr>
<td>ß</td>
<td>%ss</td>
</tr>
<tr>
<td>ø</td>
<td>%D</td>
</tr>
<tr>
<td>@</td>
<td>%at</td>
</tr>
<tr>
<td>CE</td>
<td>%CE</td>
</tr>
</tbody>
</table>

Characters that cannot be printed

Apart from text, you can also define certain non-printable characters for formatting purposes. Enter the special character \ before the non-printable characters.

The following formatting possibilities are available:

<table>
<thead>
<tr>
<th>Character</th>
<th>Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>Line break</td>
<td>\n</td>
</tr>
<tr>
<td>Horizontal tab</td>
<td>\t</td>
</tr>
<tr>
<td>(the tab width is permanently set to 8 characters)</td>
<td></td>
</tr>
<tr>
<td>Vertical tab</td>
<td>\v</td>
</tr>
<tr>
<td>(the tab width is permanently set to one line)</td>
<td></td>
</tr>
</tbody>
</table>
Engraving system variables

In addition to the standard characters, you can engrave the contents of certain system variables. Enter \% before the system variable.

You can also engrave the current date or time. Enter \%\texttt{time<x>}. \texttt{x} defines the format, e.g. 08 for DD.MM.YYYY (identical to the function \texttt{SYSSTR ID332}, see the User’s Manual for Conversational Programming, “Q parameter programming” chapter, “Copying system data to a string” section)

Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. \texttt{time08}.

<table>
<thead>
<tr>
<th>Character</th>
<th>Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>DD.MM.YYYY hh:mm:ss</td>
<td>%time00</td>
</tr>
<tr>
<td>D.MM.YYYY h:mm:ss</td>
<td>%time01</td>
</tr>
<tr>
<td>D.MM.YYYY h:mm</td>
<td>%time02</td>
</tr>
<tr>
<td>D.MM.YY h:mm</td>
<td>%time03</td>
</tr>
<tr>
<td>YYYYMM-DD hh:mm:ss</td>
<td>%time04</td>
</tr>
<tr>
<td>YYYYMM-DD hh:mm</td>
<td>%time05</td>
</tr>
<tr>
<td>YYYYMM-DD h:mm</td>
<td>%time06</td>
</tr>
<tr>
<td>YY-MM-DD h:mm</td>
<td>%time07</td>
</tr>
<tr>
<td>DD.MM.YYYY</td>
<td>%time08</td>
</tr>
<tr>
<td>D.MM.YYYY</td>
<td>%time09</td>
</tr>
<tr>
<td>D.MM.YY</td>
<td>%time10</td>
</tr>
<tr>
<td>YYYYMM-MM-DD</td>
<td>%time11</td>
</tr>
<tr>
<td>YY-MM-DD</td>
<td>%time12</td>
</tr>
<tr>
<td>hh:mm:ss</td>
<td>%time13</td>
</tr>
<tr>
<td>h:mm:ss</td>
<td>%time14</td>
</tr>
<tr>
<td>h:mm</td>
<td>%time15</td>
</tr>
</tbody>
</table>
11.9 FACE MILLING (Cycle 232, DIN/ISO: G232)

Cycle run

Cycle 232 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: Line-by-line machining, retraction and stepover at the positioning feed rate

1. From the current position, the TNC positions the tool at rapid traverse \( F_{\text{MAX}} \) to the starting position using positioning logic
   1. If the current position in the spindle axis is greater than the 2nd set-up clearance, the control positions the tool first in the machining plane and then in the spindle axis. Otherwise it first moves to the 2nd set-up clearance and then in the machining plane. The starting point in the machining plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2. The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

**Strategy Q389=0**

3. The tool subsequently advances to the stopping point 2 at the programmed feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.
4. The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5. The tool then moves back in the direction of the starting point 1.
6. The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7. In order to avoid non-productive motions, the surface is then machined in reverse direction.
8. The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9. At the end of the cycle, the tool is retracted in \( F_{\text{MAX}} \) to the 2nd set-up clearance.
### Strategy Q389=1

3. The tool subsequently advances to the end point 2 at the programmed feed rate for milling. The end point lies at the edge of the surface. The TNC calculates the end point from the programmed starting point, the programmed length and the tool radius.

4. The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.

5. The tool then moves back in the direction of the starting point 1. The motion to the next line again occurs at the edge of the workpiece.

6. The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

7. In order to avoid non-productive motions, the surface is then machined in reverse direction.

8. The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9. At the end of the cycle, the tool is retracted in $F_{\text{MAX}}$ to the 2nd set-up clearance.

### Strategy Q389=2

3. The tool subsequently advances to the stopping point 2 at the programmed feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.

4. The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at the pre-positioning feed rate directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.

5. The tool then returns to the current infeed depth and moves in the direction of the next end point 2.

6. The multipass process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

7. In order to avoid non-productive motions, the surface is then machined in reverse direction.

8. The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9. At the end of the cycle, the tool is retracted in $F_{\text{MAX}}$ to the 2nd set-up clearance.
Please note while programming:

Enter the **2nd set-up clearance** in Q204 so that no collision with the workpiece or the fixtures can occur. If the starting point in the 3rd axis Q227 and the end point in the 3rd axis Q386 are entered as equal values, the TNC does not run the cycle (depth = 0 has been programmed).

Program Q227 greater than Q386. Otherwise, the TNC will display an error message.
11.9 FACE MILLING (Cycle 232, DIN/ISO: G232)

Cycle parameters

- **Machining strategy (0/1/2) Q389**: Determine how the TNC should machine the surface:
  - 0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  - 1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
  - 2: Line-by-line machining, retraction and stepover at the positioning feed rate

- **Starting point in 1st axis Q225 (absolute)**: Starting point coordinate of the surface to be machined in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 2nd axis Q226 (absolute)**: Starting point coordinate of the surface to be machined in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **Starting point in 3rd axis Q227 (absolute)**: Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999

- **End point in 3rd axis Q386 (absolute)**: Coordinate in the spindle axis to which the surface is to be face milled. Input range -99999.9999 to 99999.9999

- **1st side length Q218 (incremental value)**: Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path in reference to the starting point in the 1st axis. Input range -99999.9999 to 99999.9999

- **2nd side length Q219 (incremental value)**: Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the starting point in the 2nd axis. Input range -99999.9999 to 99999.9999

- **Maximum plunging depth Q202 (incremental value)**: Maximum amount that the tool is advanced each time. The TNC calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range 0 to 99999.9999

- **Allowance for floor Q369 (incremental)**: Distance used for the last infeed. Input range 0 to 99999.9999
Max. path overlap factor Q370: Maximum stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. tooth radius when using a face-milling cutter), the TNC reduces the stepover accordingly. Input range 0.1 to 1.9999

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

Feed rate for finishing Q385: Traversing speed of the tool in mm/min, while milling the last infeed. Input range 0 to 99999.9999; alternatively FAUTO, FU, FZ

Feed rate for pre-positioning Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207. Input range 0 to 99999.9999, alternatively FMAX, FAUTO

Set-up clearance Q200 (incremental): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the TNC moves the tool at the set-up clearance over the current plunging depth to the starting point of the next pass. Input range 0 to 99999.9999

Clearance to side Q357 (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used. Input range 0 to 99999.9999

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

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>71 CYCL DEF 232 FACE MILLING</td>
</tr>
<tr>
<td>Q389=2 ;STRATEGY</td>
</tr>
<tr>
<td>Q225=+10 ;STARTNG PNT 1ST AXIS</td>
</tr>
<tr>
<td>Q226=+12 ;STARTNG PNT 2ND AXIS</td>
</tr>
<tr>
<td>Q227=+2.5 ;STARTNG PNT 3RD AXIS</td>
</tr>
<tr>
<td>Q386=3 ;END POINT 3RD AXIS</td>
</tr>
<tr>
<td>Q218=150 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q219=75 ;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q202=2 ;MAX. PLUNGING DEPTH</td>
</tr>
<tr>
<td>Q369=0.5 ;ALLOWANCE FOR FLOOR</td>
</tr>
<tr>
<td>Q370=1 ;MAX. TOOL PATH OVERLAP</td>
</tr>
<tr>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
</tr>
<tr>
<td>Q385=800 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q253=2000 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q200=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q357=2 ;CLEARANCE TO SIDE</td>
</tr>
<tr>
<td>Q204=2 ;2ND SET-UP CLEARANCE</td>
</tr>
</tbody>
</table>
Cycles: Special Functions

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

Cycle run

The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of table axes. Option 143 LAC (Load Adaptive Control) and Cycle 239 ASCERTAIN THE LOAD enable the control to automatically ascertain and adapt the current mass moment of inertia of the load as well as the current friction forces, or to reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The TNC performs a weighing procedure to ascertain the weight acting on the axes. In the weighing procedure, the axes move a specified distance—the machine tool builder defines the exact scope of axis movement. Before weighing, the axes are moved to a position, if required, where there is no danger of collision during the weighing procedure. This safe position is defined by the machine tool builder.

Parameter Q570 = 0

1. There is no physical movement of the axes.
2. The TNC resets the LAC.
3. The TNC activates feedforward parameters and, if applicable, controller parameters that ensure a safe movement of the axes concerned, regardless of the load condition—the parameters set with Q570=0 are independent of the current load.
4. These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

1. The TNC performs a weighing procedure in which it moves one or more axes. Which axes are moved depends on the setup of the machine and on the drives of the axes.
2. The scope of axis movement is defined by the machine tool builder.
3. The feedforward and controller parameters determined by the TNC depend on the current load.
4. The TNC activates the parameters determined.
ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, software option 11.10 143)

Please note while programming:

- Cycle 239 becomes effective immediately after definition.
  If you are using the mid-program startup function and the TNC skips Cycle 239 in the block scan, the TNC will ignore this cycle—no weighing procedure will be performed.

- The machine must be prepared by the machine tool builder for this cycle.
  Cycle 239 can only be used with option 143 LAC (Load Adaptive Control).

- This cycle may lead to extensive motion in one or more axes!
  The TNC moves the axes at rapid traverse.
  Set the potentiometer for feed-rate and rapid-traverse override to at least 50 % to ensure a correct ascertainment of the load.
  Before cycle start, the TNC might move to a safe position; this position is defined by the machine tool builder!
  Before using this cycle, contact your machine tool builder for details on the type and scope of movements performed in Cycle 239!

Cycle parameters

- **ASCERTAIN THE LOAD** Q570: Define whether the TNC is to perform the LAC (Load Adaptive Control) weighing procedure or reset the last determined load-dependent feedforward and controller parameters:
  0: Reset LAC; the values last set by the TNC are reset; the TNC uses load-independent feedforward and controller parameters
  1: Perform the weighing procedure; the TNC moves the axes to determine the feedforward and controller parameters with respect to the current load; the determined values are activated immediately

NC blocks

<table>
<thead>
<tr>
<th>62 CYCL DEF 239 ASCERTAIN THE LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q570=0 ;ASCERTAIN THE LOAD</td>
</tr>
</tbody>
</table>
11.11 Programming examples

Example: Interpolation Turning Cycle 291

Cycle 291 COUPLING TURNING INTERPOLATION is used in the following program. This programming example illustrates the machining of an axial recess and a radial recess.

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

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

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

0 BEGIN PGM 1 MM
1 BLK FORM CYLINDER Z R15 L60 Definition of workpiece blank: Cylinder
2 TOOL CALL 10 Tool call: tool for axial recess
3 CC X+0 Y+0 Retract the tool
4 LP PR+30 PA+0 R0 FMAX Position the tool in the working plane
5 CYCL DEF 291 COUPLG. TURNG. INTERP. Activate interpolation turning
   Q560=+1 ;SPINDLE COUPLING
   Q336=+0 ;ANGLE OF SPINDLE
   Q216=+0 ;CENTER IN 1ST AXIS
   Q217=+0 ;CENTER IN 2ND AXIS
   Q561=+1 ;TURNING TOOL CONVERSION
6 CYCL CALL Call the cycle
7 LP PR+9 PA+0 RR FMAX Position the tool in the working plane
8 L Z+10 FMAX Position the tool in the spindle axis
9 L Z+0.2 F2000 Recessing on face, infeed: 0.2 mm, depth: 6 mm
10 LBL 1
11 CP IPA+360 IZ-0.2 DR+ F10000
12 CALL LBL 1 REP 30
<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>LBL 2</td>
<td>Retract from recess, step: 0.4 mm</td>
</tr>
<tr>
<td>14</td>
<td>CP IPA+360 IZ+0.4 DR+</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>CALL LBL 2 REP15</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>L Z+200 R0 FMAX</td>
<td>Retract to clearance height, deactivate radius compensation</td>
</tr>
<tr>
<td>17</td>
<td>CYCL DEF 291 COUPLG. TURNG. INTERP.</td>
<td>Terminate interpolation turning</td>
</tr>
<tr>
<td></td>
<td>Q560=+0</td>
<td>;SPINDLE COUPLING</td>
</tr>
<tr>
<td></td>
<td>Q336=+0</td>
<td>;ANGLE OF SPINDLE</td>
</tr>
<tr>
<td></td>
<td>Q216=+0</td>
<td>;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td></td>
<td>Q217=+0</td>
<td>;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td></td>
<td>Q561=+0</td>
<td>;TURNING TOOL CONVERSION</td>
</tr>
<tr>
<td>18</td>
<td>CYCL CALL</td>
<td>Call the cycle</td>
</tr>
<tr>
<td>19</td>
<td>TOOL CALL 11</td>
<td>Tool call: tool for radial recess</td>
</tr>
<tr>
<td>20</td>
<td>CC X+0 Y+0</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>LP PR+25 PA+0 R0 FMAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>22</td>
<td>CYCL DEF 291 COUPLG. TURNG. INTERP.</td>
<td>Activate interpolation turning</td>
</tr>
<tr>
<td></td>
<td>Q560=+1</td>
<td>;SPINDLE COUPLING</td>
</tr>
<tr>
<td></td>
<td>Q336=+0</td>
<td>;ANGLE OF SPINDLE</td>
</tr>
<tr>
<td></td>
<td>Q216=+0</td>
<td>;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td></td>
<td>Q217=+0</td>
<td>;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td></td>
<td>Q561=+1</td>
<td>;TURNING TOOL CONVERSION</td>
</tr>
<tr>
<td>23</td>
<td>CYCL CALL</td>
<td>Call the cycle</td>
</tr>
<tr>
<td>24</td>
<td>LP PR+15.2 PA+0 RR FMAX</td>
<td>Position the tool in the working plane</td>
</tr>
<tr>
<td>25</td>
<td>L Z+10 FMAX</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>L Z-11 F7000</td>
<td>Position the tool in the spindle axis</td>
</tr>
<tr>
<td>27</td>
<td>LBL 3</td>
<td>Recessing on lateral surface, infeed: 0.2 mm, depth: 6 mm</td>
</tr>
<tr>
<td>28</td>
<td>CC X+0.1 Y+0</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>CP IPA+180 DR+ F10000</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>CC X-0.1 Y+0</td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>CP IPA+180 DR+</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>CALL LBL 3 REP15</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>LBL 4</td>
<td>Retract from recess, step: 0.4 mm</td>
</tr>
<tr>
<td>34</td>
<td>CC X-0.2 Y+0</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>CP IPA+180 DR+</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>CC X+0.2 Y+0</td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>CP IPA+180 DR+</td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>CALL LBL 4 REP8</td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>LP PR+50 FMAX</td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>L Z+200 R0 FMAX</td>
<td>Retract to clearance height, deactivate radius compensation</td>
</tr>
<tr>
<td>41</td>
<td>CYCL DEF 291 COUPLG. TURNG. INTERP.</td>
<td>Terminate interpolation turning</td>
</tr>
<tr>
<td></td>
<td>Q560=+0</td>
<td>;SPINDLE COUPLING</td>
</tr>
<tr>
<td></td>
<td>Q336=+0</td>
<td>;ANGLE OF SPINDLE</td>
</tr>
<tr>
<td></td>
<td>Q216=+0</td>
<td>;CENTER IN 1ST AXIS</td>
</tr>
</tbody>
</table>
Cycles: Special Functions

11.11 Programming examples

```plaintext
Q217=+0 ;CENTER IN 2ND AXIS
Q561=+0 ;TURNING TOOL CONVERSION

42 CYCL CALL
Call the cycle

43 TOOL CALL 11
Repeated TOOL CALL in order to override the conversion of parameter Q561

44 M30

45 END PGM 1 MM
```

Example: Interpolation Turning Cycle 292

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

**Program sequence**
- Tool call: Milling cutter D20
- Cycle 32 Tolerance
- Reference to the contour with Cycle 14
- Cycle 292 Contour turning interpolation

```plaintext
0 BEGIN PGM 2 MM

1 BLK FORM CYLINDER Z R25 L40
Definition of workpiece blank: Cylinder

2 TOOL CALL "D20" Z S111
Tool call: End mill D20

3 CYCL DEF 32.0 TOLERANCE
Use Cycle 32 to define the tolerance

4 CYCL DEF 32.1 T0.05

5 CYCL DEF 32.2 HSC-MODE:1

6 CYCL DEF 14.0 CONTOUR
Use Cycle 14 to refer to the contour in LBL1

7 CYCL DEF 14.1 CONTOUR LABEL1

8 CYCL DEF 292 CONTOUR. TURNG. INTRP.
Define Cycle 292

Q560=+1 ;SPINDLE COUPLING
Q336=+0 ;ANGLE OF SPINDLE
Q546=+3 ;CHANGE TOOL DIRECTN.
Q529=+0 ;MACHINING OPERATION
Q221=+0 ;SURFACE OVERSIZE
Q441=+1 ;INFEED
Q449=+15000 ;FEED RATE
Q491=+15 ;CONTOUR START RADIUS
Q357=+2 ;CLEARANCE TO SIDE
Q445=+50 ;CLEARANCE HEIGHT

9 L Z+50 R0 FMAX M3
Pre-position in the tool axis, spindle on

10 L X+0 Y+0 R0 FMAX M99
Pre-position in the working plane to the center of rotation, call the cycle

11 LBL 1
LBL1 contains the contour
```
### Programming examples 11.11

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>L Z+2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>X+15</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>L Z-5</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>L Z-7</td>
<td></td>
</tr>
<tr>
<td></td>
<td>X+19</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>RND R3</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>L Z-15</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>RND R2</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>L X+27</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>LBL 0</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>M30</td>
<td>End of program</td>
</tr>
<tr>
<td>21</td>
<td>END PGM 2 MM</td>
<td></td>
</tr>
</tbody>
</table>
12

Cycles: Turning
12 Cycles: Turning

12.1 Turning Cycles (software option 50)

### Overview

Defining turning cycles:
- The soft-key row shows the available groups of cycles
- Select the menu for cycle group **TURNING**
- Select cycle group, e.g. cycles for longitudinal turning
- Select cycle, e.g. TURN SHOULDER, LONGITUDINAL

The TNC offers the following cycles for turning operations:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle group</th>
<th>Cycle name</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>SPECIAL CYCLES</td>
<td>Special cycles</td>
<td>ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800)</td>
<td>332</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801)</td>
<td>338</td>
</tr>
<tr>
<td></td>
<td></td>
<td>GEAR HOBBING (Cycle 880, DIN/ISO: G880)</td>
<td>435</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)</td>
<td>440</td>
</tr>
<tr>
<td>LONGITONDAL TURNING</td>
<td>Cycles for longitudinal turning</td>
<td>TURN SHOULDER LONGITUDINAL (Cycle 811, DIN/ISO: G811)</td>
<td>339</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, DIN/ISO: G812)</td>
<td>340</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TURN, LONGITUDINAL PLUNGE (Cycle 813, DIN/ISO: G813)</td>
<td>343</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)</td>
<td>347</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G810)</td>
<td>350</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)</td>
<td>354</td>
</tr>
<tr>
<td>Soft key</td>
<td>Cycle group</td>
<td>Cycle</td>
<td>Page</td>
</tr>
<tr>
<td>----------</td>
<td>-----------------------------</td>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td></td>
<td><strong>Cycles for transverse turning</strong></td>
<td><strong>TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)</strong></td>
<td>362</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)</strong></td>
<td>365</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>TURN, TRANSVERSE PLUNGE (Cycle 823, DIN/ISO: G823)</strong></td>
<td>369</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, DIN/ISO: G824)</strong></td>
<td>372</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>TURN CONTOUR FACE (Cycle 820, DIN/ISO: G820)</strong></td>
<td>376</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>TURN CONTOUR-PARALLEL (Cycle 815, DIN/ISO: G815)</strong></td>
<td>358</td>
</tr>
<tr>
<td></td>
<td><strong>Cycles for recessing</strong></td>
<td><strong>SIMPLE RADIAL RECESSING (Cycle 841, DIN/ISO: G841)</strong></td>
<td>380</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>RADIAL RECESSING EXTENDED (Cycle 842, DIN/ISO: G842)</strong></td>
<td>383</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>RECESSING CONTOUR RADIAL (Cycle 840, DIN/ISO: G840)</strong></td>
<td>388</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>SIMPLE AXIAL RECESSING (Cycle 851, DIN/ISO: G851)</strong></td>
<td>392</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>AXIAL RECESSING EXTENDED (Cycle 852, DIN/ISO: G852)</strong></td>
<td>395</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>AXIAL RECESSING (Cycle 850, DIN/ISO: G850)</strong></td>
<td>400</td>
</tr>
</tbody>
</table>
## Cycles: Turning

### 12.1 Turning Cycles (software option 50)

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle group</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cycles for recessing</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G061</td>
<td></td>
<td>RADIAL RECESSING (Cycle 861, DIN/ISO: G861)</td>
<td>404</td>
</tr>
<tr>
<td>G062</td>
<td></td>
<td>RADIAL RECESSING EXTENDED (Cycle 862, DIN/ISO: G862)</td>
<td>407</td>
</tr>
<tr>
<td>G060</td>
<td></td>
<td>RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)</td>
<td>411</td>
</tr>
<tr>
<td>G071</td>
<td></td>
<td>AXIAL RECESSING (Cycle 871, DIN/ISO: G871)</td>
<td>415</td>
</tr>
<tr>
<td>G072</td>
<td></td>
<td>AXIAL RECESSING EXTENDED (Cycle 872, DIN/ISO: G872)</td>
<td>417</td>
</tr>
<tr>
<td>G070</td>
<td></td>
<td>AXIAL RECESSING (Cycle 870, DIN/ISO: G870)</td>
<td>421</td>
</tr>
<tr>
<td></td>
<td>Cycles for thread turning</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G031</td>
<td></td>
<td>THREAD LONGITUDINAL (Cycle 831, DIN/ISO: G831)</td>
<td>424</td>
</tr>
<tr>
<td>G032</td>
<td></td>
<td>THREAD EXTENDED (Cycle 832, DIN/ISO: G832)</td>
<td>427</td>
</tr>
<tr>
<td>G030</td>
<td></td>
<td>CONTOUR-PARALLEL THREAD (Cycle 830, DIN/ISO: G830)</td>
<td>431</td>
</tr>
</tbody>
</table>
Working with turning cycles

You can only use turning cycles in Turning mode
FUNCTION MODE TURN.

In turning cycles the TNC takes into account the cutting geometry (TO, RS, P-ANGLE, T-ANGLE) of the tool so that damage to the defined contour elements is prevented. The TNC outputs a warning if complete machining of the contour with the active tool is not possible.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the TNC detects the machining position (inside/outside machining) via the starting position or tool position when the cycle is called. In some cycles you can also enter the machining position directly in the cycle. After modifying the machining position, check the tool position and rotation direction.

If you program M136 before a cycle, the TNC interprets feed rate values in the cycle in mm/rev., and without M136 in mm/min.

If turning cycles are executed during inclined machining (M144), the angles of the tool to the contour change. The TNC automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

Some cycles machine contours that you have written in a subprogram. You program these contours with plain-language path functions or FK functions. Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.

You must call turning cycles 880 and 81x - 87x with CYCL CALL or M99. Before calling a cycle, be sure to program:

- Turning mode FUNCTION MODE TURN
- Tool call TOOL CALL
- Direction of rotation of turning spindle, e.g. M303
- Selection of speed/cutting speed FUNCTION TURNDATA SPIN
- If you use feed rate per revolution mm/rev., M136
- Tool positioning to suitable starting point e.g. L X+130 Y+0 R0 FMAX
- Adaptation of coordinate system and align tool CYCL DEF 800 ADAPT ROTARY COORDINATE SYSTEM
12 Cycles: Turning

12.1 Turning Cycles (software option 50)

Blank form update (FUNCTION TURNDATA)

During turning operations workpieces must often be machined with several tools. Often a contour element cannot be completely finished because the tool form does not permit this (e.g. with a back cut). In this case, single sub-areas have to be reworked with other tools. The TNC detects the already machined areas with the blank form update and adapts all approach and departure paths to the specific, current machining situation. With the shorter machining paths, traverses in the air are avoided to significantly reduce machining time.

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

The TNC optimizes machining areas and approach motions with blank form update. The TNC takes into account the specific tracked workpiece blank for approach and departure paths. If parts of the finished part extend beyond the workpiece blank, this may damage the workpiece and tool.

Blank form update is only possible with cycle machining in turning mode (FUNCTION MODE TURN).

You must define a closed contour as the workpiece blank for the blank form update (start position = end position). The workpiece blank corresponds to the cross-section of a rotationally symmetrical body.
The TNC has various options for defining the blank:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Workpiece blank definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>BLANK OFF</td>
<td>Deactivate blank form update</td>
</tr>
<tr>
<td>BLANK &lt;FILE&gt;</td>
<td>Workpiece blank definition in a program: Enter name of the file</td>
</tr>
<tr>
<td>BLANK &lt;FILE&gt;+HS</td>
<td>Workpiece blank definition in a program: Enter string parameter with the program name</td>
</tr>
<tr>
<td>BLANK LBL NR</td>
<td>Workpiece blank definition in a subprogram: Enter number of the subprogram</td>
</tr>
<tr>
<td>BLANK LBL NAME</td>
<td>Workpiece blank definition in a subprogram: Enter name of the subprogram</td>
</tr>
<tr>
<td>BLANK LBL OS</td>
<td>Workpiece blank definition in a subprogram: Enter string parameter with the subprogram name</td>
</tr>
</tbody>
</table>

Activate blank form update and define workpiece blank:

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

**NC syntax**

```
11 FUNCTION TURNDATABLANK LBL 20
```
Cycles: Turning

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

Application

This function must be adapted to the TNC by your machine manufacturer. Refer to your machine manual.

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

The angle of incidence between the tool and the turning spindle is important for turning operations, for example, in order to machine contours with undercuts. Cycle 800 provides various possibilities for aligning the coordinate system for an inclined machining operation:

- If you have already positioned the tilting axis for an inclined machining operation, you can use Cycle 800 to align the coordinate system relative to the position of the tilting axes (Q530=0).
- Cycle 800 uses the angle of incidence Q531 to calculate the required tilting axis angle. Depending on the strategy selected in parameter INCLINED MACHINING Q530, the TNC positions the tilting axis with (Q530=1) or without compensating movement (Q530=2).
- Cycle 800 uses the angle of incidence Q531 to calculate the required tilting axis angle, but does not perform any movements for positioning the tilting axis (Q530=3). You need to position the tilting axis to the calculated values Q120 (A axis), Q121 (B axis) and Q122 (C axis) after the cycle.

If you change the position of a tilting axis, you need to run Cycle 800 again to align the coordinate system.
ADAPT ROTARY COORDINATE SYSTEM  12.2
(Cycle 800, DIN/ISO: G800)

If the axis of the milling spindle and the axis of the turning spindle are aligned parallel to each other, you can use **PRECESSION ANGLE Q497** to define any desired rotation of the coordinate system around the spindle axis (Z axis). This may be necessary if you have to bring the tool into a specific position due to space restrictions or if you want to improve your ability to observe a machining process. If the axis of the turning spindle is not aligned parallel to the axis of the milling spindle, only two precession angles are useful for machining. The TNC selects the angle that is closest to the input value Q497.

Cycle 800 positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can also use the mirrored tool (**REVERSE TOOL Q498**), thereby moving the position of the milling spindle by 180°. In this way you can use a tool for both inside and outside machining. Position the cutting edge at the center of the turning spindle using a positioning block, such as **L Y +0 R0 FMAX**.
Cycles: Turning

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

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of rotation is aligned with the axis of the turning spindle (e.g. if large or rotationally non-symmetrical workpieces are being used). The Q535 eccentric turning function in Cycle 800 enables you to perform turning operations in such cases as well.

During eccentric turning more than one linear axis is coupled to the turning spindle. The TNC compensates the eccentricity by performing circular compensating movements with the coupled linear axes.

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

High rotational speeds and large eccentricity require high feed rates of the linear axes to ensure that they move synchronously. If these feed rates are not maintained, the contour will be damaged. The TNC therefore generates an error message if 80% of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the rotational speed.

Do not couple or decouple axes when the turning spindle is rotating. The TNC performs compensating movements during coupling and decoupling. Check for possible collisions.

Turn a test cut before the actual machining operation to ensure that the required speeds can be attained. The linear axis positions resulting from the compensation are displayed by the TNC only in the ACTUAL value position display.

The rotation of the workpiece creates centrifugal forces that can cause vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life. High centrifugal forces can damage the machine or push the workpiece out of the fixture.

Danger of collision!
Collision monitoring (DCM) is not active during eccentric turning. The TNC displays a corresponding warning during eccentric turning.
Effect

With Cycle 800 ADAPT ROTARY COORDINATE SYSTEM, the TNC aligns the workpiece coordinate system and orients the tool correspondingly. Cycle 800 is effective until it is reset by Cycle 801, or until Cycle 800 is defined again. Some cycle functions of Cycle 800 are additionally reset by other factors:

- The mirroring of the tool data (Q498 REVERSE TOOL) is reset by a TOOL CALL.
- The ECCENTRIC TURNING Q535 function is reset at the end of the program or due to cancelation of the program (internal stop).

Please note while programming:

The Cycle 800 ADAPT ROTARY COORDINATE SYSTEM is machine-dependent. Refer to your machine manual.
Software option 50 must be enabled.

The tool must be clamped and measured in the correct position.
You can only mirror the tool data (Q498 REVERSE TOOL) when a turning tool is selected.
Check the orientation of the tool before machining.
Cycle 800 limits the maximum spindle speed during eccentric turning. Therefore, program Cycle 801 to reset Cycle 800, and reset the speed limitation with FUNCTION TURNDATA SPIN SMAX.
If you use the settings 1: MOVE, 2: TURN and 3: STAY in the parameter Q530 INCLINED MACHINING, the TNC activates the function M144 (see also User’s Manual for Inclined Turning).
Cycles: Turning

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

Cycle parameters

- **PRECESSION ANGLE** Q497: Angle to which the TNC aligns the tool. Input range 0 to 359.9999
- **REVERSE TOOL** Q498: mirror tool for inside/ outside machining. Input range 0 and 1.
- **Inclined machining** Q530: Position the tilting axes for inclined machining:
  0: The position of the tilting axis (axis must have been positioned before) remains unchanged
  1: Position the tilting axis automatically, thereby orienting the tool tip (MOVE). The relative position between the tool and workpiece remains unchanged. The TNC performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
  3: Do not position the tilting axis. Position the tilting axes in a subsequent and separate positioning block (STAY). The TNC stores the position values in the parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis).
- **Angle of incidence** Q531: Angle of incidence for aligning the tool. Input range: -180° to +180°
- **Feed rate for positioning** Q532: Traverse speed of the tilting axis during automatic positioning. Input range 0.001 to 99999.999
- **Preferred direction** Q533: Selection of alternate possibilities of inclination. The angle of incidence you have defined is used by the TNC to calculate the appropriate positioning of the tilting axes present on the machine. In general there are always two possible solutions. Using parameter Q533, configure which of the solution options the TNC should apply:
  0: Choose option with shortest way
  -1: Choose option with negative direction
  +1: Choose option with positive direction
  -2: Choose option with negative direction at a range of between -90° and -180°
  +2: Choose option with positive direction at a range of between +90° and +180°
**Eccentric turning** Q535: Couple the axes for the eccentric turning operation:

0: Deactivate axis couplings

1: Activate axis couplings. The center of rotation is located at the active preset

2: Activate axis couplings. The center of rotation is located at the active datum

3: Do not change axis couplings.

**Eccentric turning without stop** Q536: Interrupt program run before the axes are coupled:

0: Stop before the axes are coupled again. In stopped condition, the TNC opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. Then press NC start to continue machining or press the **CANCEL** soft key to cancel machining

1: Axes are coupled without stopping beforehand
12.3  RESET ROTARY COORDINATE SYSTEM  
(Cycle 801, DIN/ISO: G801)

Please note while programming:

- The Cycle 801 RESET ROTARY COORDINATE SYSTEM is machine-dependent. Refer to your machine manual.

With Cycle 801 RESET ROTARY COORDINATE SYSTEM you can reset the settings you have made with Cycle 800 ADAPT ROTARY COORDINATE SYSTEM. Cycle 800 limits the maximum spindle speed during eccentric turning. Therefore, program Cycle 801 to reset Cycle 800, and reset the speed limitation with FUNCTION TURNDATA SPIN SMAX.

Effect
Cycle 801 resets the following settings you have programmed with Cycle 800:
- Precession angle Q497
- Reverse tool Q498

If you have executed the eccentric turning function with Cycle 800, the cycle limits the maximum spindle speed. To reset this, program FUNCTION TURNDATA SPIN SMAX in addition to Cycle 801.

- Cycle 801 does not orient the tool to the starting position. If a tool was oriented with Cycle 800, it remains in this position also after resetting.

Cycle parameters

- Cycle 801 does not have a cycle parameter. Finish the cycle input with the “END” key.
12.4 Fundamentals of Turning Cycles

The pre-positioning of the tool decisively affects the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when a cycle is called. When calculating the area to be machined, the TNC takes into account the starting point and the end point defined in the cycle or contour defined in the cycle. If the starting point lies in the area to be machined the TNC positions the tool beforehand in some cycles to set-up clearance.

The turning direction with 81x cycles is longitudinal to the rotary axis and lateral to the rotary axis with 82x cycles. The motions are contour-parallel in cycle 815.

The cycles can be used for inside and outside machining. The TNC takes the information for this from the position of the tool or the definition in the cycle (see “Working with turning cycles”, page 329).

In cycles with freely defined contours (Cycles 810, 820 and 815), the programming direction of the contour determines the direction of machining.

In cycles for turning you can specify the machining strategies of roughing, finishing or complete machining.

Caution: Danger to the workpiece and tool!

The turning cycles position the tool automatically to the starting point during finishing. The approach strategy is influenced by the position of the tool when the cycle is called. The decisive factor is whether the tool is located inside or outside an envelope contour when the cycle is called. The envelope contour is the programmed contour, enlarged by the set-up clearance.

If the tool is within the envelope contour, the cycle positions the tool at the defined feed rate directly to the starting position. This can cause contour damage. Position the tool at a sufficient distance from the starting point to prevent the possibility of contour damage.

If the tool is outside the envelope contour, positioning to the envelope contour is performed at rapid traverse, and at the programmed feed rate within the envelope contour.
Cycles: Turning

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

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

Application
This cycle enables you to carry out longitudinal turning of right-angled shoulders.
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run
The cycle processes the area from the tool position to the end point defined in the cycle.
1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with Q463 MAX. CUTTING DEPTH.
2 The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate Q478.
3 The TNC returns the tool at the defined feed rate by one infeed value.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run
1 The TNC traverses the tool in the Z coordinate by the set-up clearance $Q_{460}$. The movement is performed at rapid traverse.
2 The TNC runs the paraxial infeed motion at rapid traverse.
3 The TNC finishes the finished part contour at the defined feed rate $Q_{505}$.
4 The TNC returns the tool to set-up clearance at the defined feed rate.
5 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation $R_0$ before the cycle call.
The tool position at cycle call defines the size of the area to be machined (cycle starting point).
Also refer to the fundamentals of turning cycles (see page 339).
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

### NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 811 TURN SHOULDER LONG.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q493=+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=+55 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.6 TURN SHOULDERS LONGITUDINAL EXTENDED

(Cycle 812, DIN/ISO: G812)

Application

This cycle enables you to run longitudinal turning of shoulders.

Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle, you can define angles for the face and circumferential surfaces.
- You can insert a radius in the contour edge.

You can use the cycle either for roughing, finishing, or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter \( Q_{491} \) is larger than the end diameter \( Q_{493} \), the cycle runs outside machining. If the start diameter \( Q_{491} \) is less than the end diameter \( Q_{493} \), the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the starting point is within the area to be machined, the TNC positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with \( Q_{463} \) MAX. CUTTING DEPTH.

2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate \( Q_{478} \).

3. The TNC returns the tool at the defined feed rate by one infeed value.

4. The TNC positions the tool back at rapid traverse to the beginning of cut.

5. The TNC repeats this process (1 to 4) until the final contour is completed.

6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

Finishing cycle run
If the starting point lies in the area to be machined, the TNC positions the tool beforehand to set-up clearance in the Z coordinate.

1. The TNC runs the paraxial infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation R0 before the cycle call. The tool position at cycle call (cycle starting point) affects the area to be machined. Also refer to the fundamentals of turning cycles (see page 339).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the contour starting point

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of circumferential surface** Q495: Angle between the circumferential surface and the rotary axis
## 12.6 TURN SHOULDER LONGITUDINAL EXTENDED
(Cycle 812, DIN/ISO: G812)

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of starting element** Q502: Size of the starting element (chamfer section)

- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.

- **Angle of face** Q496: Angle between the face and the rotary axis

- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of end element** Q504: Size of the end element (chamfer section)

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 812 TURN SHOULDER LONG. EXTENDED.</td>
</tr>
<tr>
<td>Q215++0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460++2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491++75 ;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492++0 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493++50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494++55 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495++5 ;ANGLE OF CIRCUM. SURFACE</td>
</tr>
<tr>
<td>Q501++1 ;TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502++0.5 ;SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q500++1.5 ;RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>Q496++0 ;ANGLE OF FACE</td>
</tr>
<tr>
<td>Q503++1 ;TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>Q504++0.5 ;SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>Q463++3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478++0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483++0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484++0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505++0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506++0 ;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.7 TURN, LONGITUDINAL PLUNGE
(Cycle 813, DIN/ISO: G813)

Application
This cycle enables you to run longitudinal turning of shoulders with plunge elements (undercuts).
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the start diameter $Q491$ is larger than the end diameter $Q493$, the cycle runs outside machining. If the start diameter $Q491$ is less than the end diameter $Q493$, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than $Q492$ CONTOUR START IN Z, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.
In undercutting the TNC runs the infeed with feed rate $Q478$. The return movements are then each at set-up clearance.
1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with $Q463$ MAX. CUTTING DEPTH.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate $Q478$.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

Finishing cycle run

1 The TNC runs the infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation R0 before the cycle call. The tool position at cycle call (cycle starting point) affects the area to be machined. The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC. Also refer to the fundamentals of turning cycles (see page 339).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the perpendicular to the rotary axis.

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

### NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 813 TURN, LONGITUDINAL PLUNGE</td>
</tr>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491=+75 ;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q484=+10 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493=+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=+55 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495=+70 ;ANGLE OF SIDE</td>
</tr>
<tr>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
Application

This cycle enables you to run longitudinal turning of shoulders with plunge elements (undercuts). Expanded scope of function:
- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter $Q_{491}$ is larger than the end diameter $Q_{493}$, the cycle runs outside machining. If the start diameter $Q_{491}$ is less than the end diameter $Q_{493}$, the cycle runs inside machining.

Roughing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than $Q_{492}$ CONTOUR START IN Z, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting the TNC runs the infeed with feed rate $Q_{478}$. The return movements are then each at set-up clearance.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with $Q_{463}$ MAX. CUTTING DEPTH.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate $Q_{478}$.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

- Program a positioning block to a safe position with radius compensation R0 before the cycle call.
- The tool position at cycle call (cycle starting point) affects the area to be machined.
- The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements.
- If complete machining with the active tool is not possible, a warning is output by the TNC.
- Also refer to the fundamentals of turning cycles (see page 339).
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the perpendicular to the rotary axis.
TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, DIN/ISO: G814)

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of starting element** Q502: Size of the starting element (chamfer section)

- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.

- **Angle of face** Q496: Angle between the face and the rotary axis

- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of end element** Q504: Size of the end element (chamfer section)

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 814 TURN, LONGITUDINAL PLUNGE EXT.</td>
</tr>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491=+75 ;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492=-10 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493=+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=-55 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495=+70 ;ANGLE OF SIDE</td>
</tr>
<tr>
<td>Q501=+1 ;TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502=+0.5 ;SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q500=+1.5 ;RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>Q496=+0 ;ANGLE OF FACE</td>
</tr>
<tr>
<td>Q503=+1 ;TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>Q504=+0.5 ;SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
Cycles: Turning

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

Application
This cycle enables you to run longitudinal turning of workpieces with any turning contours. The contour description is in a subprogram.
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.
1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with **Q463 MAX. CUTTING DEPTH**.
2 The TNC machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially with the defined feed rate **Q478**.
3 The TNC returns the tool at the defined feed rate by one infeed value.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate \( Q_{505} \).
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

| ! | The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called. |
| ! | Program a positioning block to a safe position with radius compensation \( R_0 \) before the cycle call. The tool position at cycle call (cycle starting point) affects the area to be machined. The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements. If complete machining with the active tool is not possible, a warning is output by the TNC. Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number. Also refer to the fundamentals of turning cycles (see page 339). When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram. |
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Reverse contour** Q499: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
TURN CONTOUR LONGITUDINAL  12.9
(Cycle 810, DIN/ISO: G810)

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Plunging** Q487: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (Q480/Q482)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

### NC blocks

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 CYCL DEF 14.0 CONTOUR</td>
<td></td>
</tr>
<tr>
<td>10 CYCL DEF 14.1 CONTOUR LABEL2</td>
<td></td>
</tr>
<tr>
<td>11 CYCL DEF 810 TURN CONTOUR LONG.</td>
<td></td>
</tr>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
<td></td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q499=+0 ;REVERSE CONTOUR</td>
<td></td>
</tr>
<tr>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
<td></td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
<td></td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
<td></td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
<td></td>
</tr>
<tr>
<td>Q487=+1 ;PLUNGE</td>
<td></td>
</tr>
<tr>
<td>Q488=+0 ;PLUNGING FEED RATE</td>
<td></td>
</tr>
<tr>
<td>Q479=+0 ;CUTTING LIMIT</td>
<td></td>
</tr>
<tr>
<td>Q480=+0 ;LIMIT VALUE FOR DIAMETER</td>
<td></td>
</tr>
<tr>
<td>Q482=+0 ;LIMIT VALUE IN Z</td>
<td></td>
</tr>
<tr>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
<td></td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL</td>
<td></td>
</tr>
<tr>
<td>14 M30</td>
<td></td>
</tr>
<tr>
<td>15 LBL 2</td>
<td></td>
</tr>
<tr>
<td>16 L X+60 Z+0</td>
<td></td>
</tr>
<tr>
<td>17 L Z-10</td>
<td></td>
</tr>
<tr>
<td>18 RND R5</td>
<td></td>
</tr>
<tr>
<td>19 L X+40 Z-35</td>
<td></td>
</tr>
<tr>
<td>20 RND R5</td>
<td></td>
</tr>
<tr>
<td>21 L X+50 Z-40</td>
<td></td>
</tr>
<tr>
<td>22 L Z-55</td>
<td></td>
</tr>
<tr>
<td>23 CC X+60 Z-55</td>
<td></td>
</tr>
<tr>
<td>24 C X+60 Z-60</td>
<td></td>
</tr>
<tr>
<td>25 L X+100</td>
<td></td>
</tr>
<tr>
<td>26 LBL 0</td>
<td></td>
</tr>
</tbody>
</table>
12.10 TURN CONTOUR-PARALLEL
(Cycle 815, DIN/ISO: G815)

Application
This cycle enables you to machine workpieces with any turning contours. The contour description is in a subprogram.
You can use the cycle either for roughing, finishing or complete machining. Turning with roughing is contour-parallel.
The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.
1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with Q463 MAX. CUTTING DEPTH.
2. The TNC machines the area between the starting position and end point. The cut is run contour-parallel with the defined feed rate Q478.
3. The TNC returns the tool at the defined feed rate back to the starting position in the X coordinate.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.
1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

- Program a positioning block to a safe position with radius compensation R0 before the cycle call.
- The tool position at cycle call (cycle starting point) affects the area to be machined.
- The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements.
- If complete machining with the active tool is not possible, a warning is output by the TNC.
- Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.
- Also refer to the fundamentals of turning cycles (see page 339).
- When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
Cycles: Turning
12.10 TURN CONTOUR-PARALLEL
(Cycle 815, DIN/ISO: G815)

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Oversize for blank** Q485 (incremental): Contour-parallel oversize for the defined contour

- **Cutting lines** Q486: Define the type of cutting lines:
  0: Cuts with constant chip cross section
  1: Equidistant proportioning of cuts

- **Reverse contour** Q499: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
TURN CONTOUR-PARALLEL  12.10
(Cycle 815, DIN/ISO: G815)

- Roughing feed rate Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- Oversize in diameter Q483 (incremental): Diameter oversize for the defined contour
- Oversize in Z Q484 (incremental): Oversize for the defined contour in axial direction
- Finishing feed rate Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 CYL DEF 14.0 CONTOUR</td>
</tr>
<tr>
<td>10 CYL DEF 14.1 CONTOUR LABEL2</td>
</tr>
<tr>
<td>11 CYL DEF 815 TURN CONTOUR-PARALLEL</td>
</tr>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q485=+5 ;OVERSIZE ON BLANK</td>
</tr>
<tr>
<td>Q486=+0 ;CUTTING LINES</td>
</tr>
<tr>
<td>Q499=+0 ;REVERSE CONTOUR</td>
</tr>
<tr>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
<tr>
<td>14 M30</td>
</tr>
<tr>
<td>15 LBL 2</td>
</tr>
<tr>
<td>16 L X+60 Z+0</td>
</tr>
<tr>
<td>17 L Z-10</td>
</tr>
<tr>
<td>18 RND R5</td>
</tr>
<tr>
<td>19 L X+40 Z-35</td>
</tr>
<tr>
<td>20 RND R5</td>
</tr>
<tr>
<td>21 L X+50 Z-40</td>
</tr>
<tr>
<td>22 L Z-55</td>
</tr>
<tr>
<td>23 CC X+60 Z-55</td>
</tr>
<tr>
<td>24 C X+60 Z-60</td>
</tr>
<tr>
<td>25 L X+100</td>
</tr>
<tr>
<td>26 LBL 0</td>
</tr>
</tbody>
</table>
Cycles: Turning

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

Application
This cycle enables you to face turn right-angled shoulders. You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing. The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

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

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with \( Q_{463} \) \text{ MAX. CUTTING DEPTH.} \)

2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate \( Q_{478} \).

3. The TNC returns the tool at the defined feed rate by one infeed value.

4. The TNC positions the tool back at rapid traverse to the beginning of cut.

5. The TNC repeats this process (1 to 4) until the final contour is completed.

6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

1. The TNC traverses the tool in the Z coordinate by the set-up clearance $Q_{460}$. The movement is performed at rapid traverse.
2. The TNC runs the paraxial infeed motion at rapid traverse.
3. The TNC finishes the finished part contour at the defined feed rate $Q_{505}$.
4. The TNC returns the tool to set-up clearance at the defined feed rate.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation $R_0$ before the cycle call.

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

Also refer to the fundamentals of turning cycles (see page 339).
Cycles: Turning
12.11 TURN SHOULDER FACE
(Cycle 821, DIN/ISO: G821)

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 821 TURN SHOULDER FACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q493=+30 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=+5 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.12 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)

Application
This cycle enables you to face turn shoulders. Expanded scope of function:
- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces.
- You can insert a radius in the contour edge.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter Q491 is larger than the end diameter Q493, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the starting point is within the area to be machined, the TNC positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with Q463 MAX. CUTTING DEPTH.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate Q478.
3. The TNC returns the tool at the defined feed rate by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning
12.12 TURN SHOULDER FACE EXTENDED
(Cycle 822, DIN/ISO: G822)

Finishing cycle run
1 The TNC runs the paraxial infeed motion at rapid traverse.
2 The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3 The TNC returns the tool to set-up clearance at the defined feed rate.
4 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation R0 before the cycle call. The tool position at cycle call (cycle starting point) affects the area to be machined. Also refer to the fundamentals of turning cycles (see page 339).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the contour starting point

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of face** Q495: Angle between the face and the rotary axis

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
Cycles: Turning

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

- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Angle of circumferential surface** Q496: Angle between the circumferential surface and the rotary axis
- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

### NC blocks

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>CYCL DEF 822 TURN SHOULDER FACE EXTENDED</td>
</tr>
<tr>
<td>Q215=+0</td>
<td>: MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2</td>
<td>: SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491=+75</td>
<td>: DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492=+0</td>
<td>: CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493=+30</td>
<td>: DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=+15</td>
<td>: CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495=+0</td>
<td>: ANGLE OF FACE</td>
</tr>
<tr>
<td>Q501=+1</td>
<td>: TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502=+0.5</td>
<td>: SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q500=+1.5</td>
<td>: RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>Q496=+5</td>
<td>: ANGLE OF CIRCUM. SURFACE</td>
</tr>
<tr>
<td>Q503=+1</td>
<td>: TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>Q504=+0.5</td>
<td>: SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>Q463=+3</td>
<td>: MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3</td>
<td>: ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4</td>
<td>: OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2</td>
<td>: OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2</td>
<td>: FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506=+0</td>
<td>: CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL</td>
<td></td>
</tr>
</tbody>
</table>
12.13 TURN, TRANSVERSE PLUNGE
(Cycle 823, DIN/ISO: G823)

Application
This cycle enables you to face turn plunge elements (undercuts). You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing. The cycle can be used for inside and outside machining. If the start diameter $Q491$ is larger than the end diameter $Q493$, the cycle runs outside machining. If the start diameter $Q491$ is less than the end diameter $Q493$, the cycle runs inside machining.

Roughing cycle run
In undercutting the TNC runs the infeed with feed rate $Q478$. The return movements are then each at set-up clearance.

1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with $Q463$ MAX. CUTTING DEPTH.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate.
3. The TNC returns the tool at the defined feed rate $Q478$ by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

Finishing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

- Program a positioning block to a safe position with radius compensation R0 before the cycle call.
- The tool position at cycle call (cycle starting point) affects the area to be machined.
- The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements.
- If complete machining with the active tool is not possible, a warning is output by the TNC.
- Also refer to the fundamentals of turning cycles (see page 339).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis

- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

---

NC blocks

<table>
<thead>
<tr>
<th>NC block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 823 TURN, TRANSVERSE PLUNGE</td>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td></td>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td></td>
<td>Q491=+75 ;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td></td>
<td>Q492=+0 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td></td>
<td>Q493=+20 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td></td>
<td>Q494=+5 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td></td>
<td>Q495=+60 ;ANGLE OF SIDE</td>
</tr>
<tr>
<td></td>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td></td>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td></td>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td></td>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td></td>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td></td>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
</tr>
</tbody>
</table>

12 L X+75 Y+0 Z+2 FMAX M303

13 CYCL CALL
Cycles: Turning

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

Application
This cycle enables you to face turn plunge elements (undercuts).
Expanded scope of function:
- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter $Q_{491}$ is larger than the end diameter $Q_{493}$, the cycle runs outside machining. If the start diameter $Q_{491}$ is less than the end diameter $Q_{493}$, the cycle runs inside machining.

Roughing cycle run
In undercutting the TNC runs the infeed with feed rate $Q_{478}$. The return movements are then each at set-up clearance.
1. The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with $Q_{463}$ MAX. CUTTING DEPTH.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate.
3. The TNC returns the tool at the defined feed rate $Q_{478}$ by one infeed value.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (1 to 4) until the final contour is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation R0 before the cycle call.
The tool position at cycle call (cycle starting point) affects the area to be machined.
The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements.
If complete machining with the active tool is not possible, a warning is output by the TNC.
Also refer to the fundamentals of turning cycles (see page 339).
Cycles: Turning
12.14 TURN, TRANSVERSE PLUNGE EXTENDED
(Cycle 824, DIN/ISO: G824)

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Diameter at contour start** Q491: X coordinate of the starting point for the plunging path (diameter value)

- **Contour start in Z** Q492: Z coordinate of the starting point for the plunging path

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
TURN, TRANSVERSE PLUNGE EXTENDED  12.14  
(Cycle 824, DIN/ISO: G824)

- **Size of starting element** Q502: Size of the starting element (chamfer section)
- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.
- **Type of end element** Q503: Define the type of element at the end of the contour (face):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Size of end element** Q504: Size of the end element (chamfer section)
- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (complete contour); retract below 45°
  2: No contour smoothing; retract below 45°

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 824 TURN, TRANSVERSE PLUNGE EXT.</td>
<td></td>
</tr>
<tr>
<td>Q215=+0</td>
<td>;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2</td>
<td>;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491=+75</td>
<td>;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492=+0</td>
<td>;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493=+20</td>
<td>;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=+10</td>
<td>;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495=+70</td>
<td>;ANGLE OF SIDE</td>
</tr>
<tr>
<td>Q501=+1</td>
<td>;TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502=+0.5</td>
<td>;SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q500=+1.5</td>
<td>;RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>Q496=+0</td>
<td>;ANGLE OF FACE</td>
</tr>
<tr>
<td>Q503=+1</td>
<td>;TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>Q504=+0.5</td>
<td>;SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>Q463=+3</td>
<td>;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3</td>
<td>;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4</td>
<td>;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2</td>
<td>;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2</td>
<td>;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q506=+0</td>
<td>;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL</td>
<td></td>
</tr>
</tbody>
</table>
12 Cycles: Turning

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

Application
This cycle enables you to face turn workpieces with any turning contours. The contour description is in a subprogram. You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing. The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to the contour starting point and begins the cycle there.
1 The TNC runs a paraxial infeed motion at rapid traverse. The infeed value is calculated by the TNC with \texttt{Q463 MAX. CUTTING DEPTH}.
2 The TNC machines the area between the starting position and the end point in traverse direction. The transverse cut is run paraxially with the defined feed rate \texttt{Q478}.
3 The TNC returns the tool at the defined feed rate by one infeed value.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC repeats this process (1 to 4) until the final contour is completed.
6 The TNC positions the tool back at rapid traverse to the cycle starting point.
**Finishing cycle run**
If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The TNC runs the infeed motion at rapid traverse.
2. The TNC finishes the finished part contour (contour starting point to contour end point) at the defined feed rate Q505.
3. The TNC returns the tool to set-up clearance at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

**Please note while programming:**

- The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.
- The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

- Program a positioning block to a safe position with radius compensation R0 before the cycle call.
- The tool position at cycle call (cycle starting point) affects the area to be machined.
- The TNC takes the cutting geometry of the tool into account to prevent damage to contour elements.
- If complete machining with the active tool is not possible, a warning is output by the TNC.
- Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.
- Also refer to the fundamentals of turning cycles (see page 339).
- When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
Cycles: Turning
12.15 TURN CONTOUR FACE
(Cycle 820, DIN/ISO: G820)

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning.

- **Reverse contour** Q499: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction

- **Maximum cutting depth** Q463: Maximum infeed in axial direction. The infeed is divided evenly to avoid abrasive cuts.

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour.
**TURN CONTOUR FACE  12.15**  
(Cycle 820, DIN/ISO: G820)

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Plunging** Q487: Permit machining of plunging elements:
  0: Do not machine plunging elements  
  1: Machine plunging elements
- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active  
  1: Cutting limit (Q480/Q482)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Contour smoothing** Q506:
  0: After each cut along the contour (within the infeed range)  
  1: Contour smoothing after the last cut (complete contour); retract below 45°  
  2: No contour smoothing; retract below 45°

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>9 CYCL DEF 14.0 CONTOUR</th>
<th>10 CYCL DEF 14.1 CONTOUR LABEL2</th>
<th>11 CYCL DEF 820 TURN CONTOUR FACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
<td>Q499=+0 ;REVERSE CONTOUR</td>
<td>Q463=+3 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
<td>Q487=+0 ;PLUNGE</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
<td>Q488=+0 ;PLUNGING FEED RATE</td>
<td>Q479=+0 ;CUTTING LIMIT</td>
<td>Q489=+0 ;CUTTING LIMIT</td>
</tr>
<tr>
<td>Q480=+0 ;LIMIT VALUE FOR DIAMETER</td>
<td>Q482=+0.2 ;LIMIT VALUE FOR DIAMETER</td>
<td>Q480=+0 ;LIMIT VALUE IN Z</td>
<td>Q482=+0 ;LIMIT VALUE IN Z</td>
</tr>
<tr>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
<td>Q480=+0 ;LIMIT VALUE IN Z</td>
<td>Q506=+0 ;CONTOUR SMOOTHING</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
<td>13 CYCL CALL</td>
<td>14 M30</td>
<td>15 LBL 2</td>
</tr>
<tr>
<td>16 L X+75 Z-20</td>
<td>17 L X+50</td>
<td>18 RND R2</td>
<td>19 L X+20 Z-25</td>
</tr>
<tr>
<td>20 RND R2</td>
<td>21 L Z+0</td>
<td>22 LBL 0</td>
<td></td>
</tr>
</tbody>
</table>
Cycles: Turning

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

Application
This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. The cycle processes only the area from the cycle starting point to the end point defined in the cycle.
1 From the cycle starting point, the TNC recesses until the first plunging depth.
2 The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate Q478.
3 If the input parameter Q488 is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction Q507=1 was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction Q507=0, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The TNC repeats this process (2 to 4) until the slot depth is reached.
7 The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8 The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate \( Q_{505} \).
3. The TNC finishes the slot floor at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate \( Q_{505} \).
7. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

- Program a positioning block to the starting position with radius compensation \( R_0 \) before the cycle call. The tool position at cycle call defines the size of the area to be machined (cycle starting point).
- From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width \( Q_{508} \) was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width – 2 * cutting radius).
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)

- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.

- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.

NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 841 RECESS TURNG. SIMPLE R.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=&gt;0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=&gt;2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q493=&gt;50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=&gt;50 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q478=&gt;0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=&gt;0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=&gt;0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=&gt;0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q463=&gt;2 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q507=&gt;0 ;MACHINING DIRECTION</td>
</tr>
<tr>
<td>Q508=&gt;0 ;OFFSET WIDTH</td>
</tr>
<tr>
<td>Q509=&gt;0 ;DEPTH COMPENSATION</td>
</tr>
<tr>
<td>Q488=&gt;0 ;PLUNGING FEED RATE</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z-25 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.17 RADIAL RECESSING EXTENDED
(Cycle 842, DIN/ISO: G842)

Application
This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:
- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter $Q491$ is larger than the end diameter $Q493$, the cycle runs outside machining. If the start diameter $Q491$ is less than the end diameter $Q493$, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than $Q491$ DIAMETER AT CONTOUR START, the TNC positions the tool in the X coordinate to $Q491$ and begins the cycle there.

1. From the cycle starting point, the TNC recesses until the first plunging depth.
2. The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate $Q478$.
3. If the input parameter $Q488$ is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction $Q507=1$ was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction $Q507=0$, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The TNC repeats this process (2 to 4) until the slot depth is reached.
7. The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than $Q491$ DIAMETER AT CONTOUR START, the TNC positions the tool in the X coordinate to $Q491$ and begins the cycle there.

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate $Q505$.
3. The TNC finishes the slot floor at the defined feed rate. If a radius for contour edges $Q500$ was specified, the TNC finishes the complete slot in one pass.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate $Q505$.
7. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

- Program a positioning block to the starting position with radius compensation $R0$ before the cycle call.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point).
- From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width $Q508$ was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutting width – 2 × cutting radius).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the contour starting point

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle between the side at the contour starting point and the perpendicular to the rotary axis
Cycles: Turning

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

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of starting element** Q502: Size of the starting element (chamfer section)

- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.

- **Angle of second side** Q496: Angle between the side at the contour end point and the perpendicular to the rotary axis

- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of end element** Q504: Size of the end element (chamfer section)

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483: Diameter oversize for the defined contour

- **Oversize in Z** Q484: Oversize in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)

- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

### NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 842 RADIAL RECESSING EXTENDED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215==0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460==2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491==75 ;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492==20 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493==50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494==50 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495==5 ;ANGLE OF SIDE</td>
</tr>
<tr>
<td>Q501==1 ;TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502==0.5 ;SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q500==1.5 ;RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>Q496==5 ;ANGLE OF SECOND SIDE</td>
</tr>
<tr>
<td>Q503==1 ;TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>Q504==0.5 ;SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>Q478==0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483==0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484==0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q463==2 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q507==0 ;MACHINING DIRECTION</td>
</tr>
<tr>
<td>Q508==0 ;OFFSET WIDTH</td>
</tr>
<tr>
<td>Q509==0 ;DEPTH COMPENSATION</td>
</tr>
<tr>
<td>Q488==0 ;PLUNGING FEED RATE</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
Turning depth compensation Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.

Feed rate for plunging Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
Cycles: Turning

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

Application
This cycle enables you to recess right-angled slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined.
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the X coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the X coordinate to the contour starting point and begins the cycle there.
1 The TNC positions the tool at rapid traverse in the Z coordinate (first cut-in position).
2 The TNC recesses until the first plunging depth.
3 The TNC cuts the area between the starting position and the end point in longitudinal direction at the defined feed rate Q478.
4 If the input parameter $Q488$ is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5 If only one machining direction $Q507=1$ was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction $Q507=0$, infeed is on both sides.
6 The tool recesses to the next plunging depth.
7 The TNC repeats this process (2 to 4) until the slot depth is reached.
8 The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side walls of the slot at the defined feed rate Q505.
3. The TNC finishes the slot floor at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

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

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

Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.

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

From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width Q508 was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width – 2 * cutting radius).
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
RECESSING CONTOUR RADIAL 12.18
(Cycle 840, DIN/ISO: G840)

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (Q480/Q482)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)
- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.
- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.
- **Reverse contour** Q499: Machining direction:
  0: Machining in contour direction
  1: Machining opposite the contour direction

### NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>CYCL DEF 14.0 CONTOUR</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>CYCL DEF 14.1 CONTOUR LABEL2</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>CYCL DEF 840 RECESS TURNG. RAD</td>
<td></td>
</tr>
<tr>
<td>Q215</td>
<td>+0</td>
<td>;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460</td>
<td>+2</td>
<td>;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q478</td>
<td>+0.3</td>
<td>;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q488</td>
<td>+0</td>
<td>;PLUNGING FEED RATE</td>
</tr>
<tr>
<td>Q483</td>
<td>+0.4</td>
<td>;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484</td>
<td>+0.2</td>
<td>;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505</td>
<td>+0.2</td>
<td>;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q479</td>
<td>+0</td>
<td>;CUTTING LIMIT</td>
</tr>
<tr>
<td>Q480</td>
<td>+0</td>
<td>;LIMIT VALUE FOR DIAMETER</td>
</tr>
<tr>
<td>Q482</td>
<td>+0</td>
<td>;LIMIT VALUE IN Z</td>
</tr>
<tr>
<td>Q463</td>
<td>+2</td>
<td>;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q507</td>
<td>+0</td>
<td>;MACHINING DIRECTION</td>
</tr>
<tr>
<td>Q508</td>
<td>+0</td>
<td>;OFFSET WIDTH</td>
</tr>
<tr>
<td>Q509</td>
<td>+0</td>
<td>;DEPTH COMPENSATION</td>
</tr>
<tr>
<td>Q499</td>
<td>+0</td>
<td>;REVERSE CONTOUR</td>
</tr>
<tr>
<td>12</td>
<td>L X+75 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>CYCL CALL</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>M30</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>LBL 2</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>L X+60 Z-10</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>L X+40 Z-15</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>RND R3</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>CR X+40 Z-35 R+30 DR+</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>RND R3</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>L X+60 Z-40</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>LBL 0</td>
<td></td>
</tr>
</tbody>
</table>
Cycles: Turning
12.19 SIMPLE AXIAL RECESSING
(Cycle 851, DIN/ISO: G851)

Application
This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

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

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. The cycle processes the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the TNC recesses until the first plunging depth.
2. The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate $Q478$.
3. If the input parameter $Q488$ is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction $Q507=1$ was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction $Q507=0$, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The TNC repeats this process (2 to 4) until the slot depth is reached.
7. The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.
**Finishing cycle run**

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes the slot floor at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC positions the tool back at rapid traverse to the cycle starting point.

**Please note while programming:**

Program a positioning block to the starting position with radius compensation **R0** before the cycle call. The tool position at cycle call defines the size of the area to be machined (cycle starting point).

From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width **Q508** was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius).
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)

- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.

- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.

NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 851 RECESS TURNING, SIMPLE AXIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q493=+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=+10 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q463=+2 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q507=+0 ;MACHINING DIRECTION</td>
</tr>
<tr>
<td>Q508=+0 ;OFFSET WIDTH</td>
</tr>
<tr>
<td>Q509=+0 ;DEPTH COMPENSATION</td>
</tr>
<tr>
<td>Q488=+0 ;PLUNGING FEED RATE</td>
</tr>
<tr>
<td>12 L X+65 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.20 AXIAL RECESSING EXTENDED
(Cycle 852, DIN/ISO: G852)

Application
This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:
- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the start diameter Q491 is larger than the end diameter Q493, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than Q492 CONTOUR START IN Z, the TNC positions the tool in the Z coordinate to Q492 and begins the cycle there.
1 From the cycle starting point, the TNC recesses until the first plunging depth.
2 The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate Q478.
3 If the input parameter Q488 is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction Q507=1 was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction Q507=0, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The TNC repeats this process (2 to 4) until the slot depth is reached.
7 The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
8 The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning
12.20 AXIAL RECESSING EXTENDED
(Cycle 852, DIN/ISO: G852)

Finishing cycle run
The TNC uses the tool position as cycle starting point when a
cycle is called. If the Z coordinate of the starting point is less than
Q492 CONTOUR START IN Z, the TNC positions the tool in the Z
coordinate to Q492 and begins the cycle there.
1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed
rate Q505.
3 The TNC finishes the slot floor at the defined feed rate. If a
radius for contour edges Q500 was specified, the TNC finishes
the complete slot in one pass.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot
side.
6 The TNC finishes the side wall of the slot at the defined feed
rate Q505.
7 The TNC positions the tool back at rapid traverse to the cycle
starting point.

Please note while programming:

Program a positioning block to the starting position
with radius compensation R0 before the cycle call.
The tool position at cycle call defines the size of the
area to be machined (cycle starting point).
From the second infeed, the TNC reduces each
further cutting traverse by 0.1 mm. This reduces
lateral pressure on the tool. If the offset width Q508
was input into the cycle, the TNC reduces the cutting
traverse by this value. After clearance roughing, the
remaining material is removed with a single cut. The
TNC generates an error message if the lateral offset
exceeds 80 % of the effective cutting width (effective
cutting width = cutting width – 2 * cutting radius).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing to finished dimension
  - 3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the contour starting point

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle between the side at the contour starting point and the parallel line to the rotary axis
Cycles: Turning
12.20 AXIAL RECESSING EXTENDED
(Cycle 852, DIN/ISO: G852)

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of starting element** Q502: Size of the starting element (chamfer section)

- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.

- **Angle of second side** Q496: Angle between the side at the contour end point and the parallel line to the rotary axis

- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of end element** Q504: Size of the end element (chamfer section)

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)

- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

---

**NC blocks**

```
11 CYCL DEF 852 RECESS TURNG. AXIAL EXTENDED
Q215=+0 ;MACHINING OPERATION
Q460=+2 ;SAFETY CLEARANCE
Q491=+75 ;DIAMETER AT CONTOUR START
Q492=-20 ;CONTOUR START IN Z
Q493=+50 ;DIAMETER AT END OF CONTOUR
Q494=-50 ;CONTOUR END IN Z
Q495=+5 ;ANGLE OF SIDE
Q501=+1 ;TYPE OF STARTING ELEMENT
Q502=+0.5 ;SIZE OF STARTING ELEMENT
Q500=+1.5 ;RADIUS OF CONTOUR EDGE
Q496=+5 ;ANGLE OF SECOND SIDE
Q503=+1 ;TYPE OF END ELEMENT
Q504=+0.5 ;SIZE OF END ELEMENT
Q478=+0.3 ;ROUGHING FEED RATE
Q484=+0.2 ;OVERSIZE IN Z
Q483=+0.4 ;OVERSIZE FOR DIAMETER
Q484=+0.2 ;OVERSIZE IN Z
Q484=+0.3 ;ROUGHING FEED RATE
Q505=+0.2 ;FINISHING FEED RATE
Q463=+2 ;MAX. CUTTING DEPTH
Q507=+0 ;MACHINING DIRECTION
Q508=+0 ;OFFSET WIDTH
Q509=+0 ;DEPTH COMPENSATION
Q488=+0 ;PLUNGING FEED RATE
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
```
- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.

- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.
12.21 AXIAL RECESSING
(Cycle 850, DIN/ISO: G850)

Application
This cycle enables you to recess right-angled slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined.
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to the contour starting point and begins the cycle there.
1 The TNC positions the tool at rapid traverse in the X coordinate (first cut-in position).
2 The TNC recesses until the first plunging depth.
3 The TNC cuts the area between the starting position and the end point in traverse direction at the defined feed rate Q478.
4 If the input parameter Q488 is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5 If only one machining direction Q507=1 was specified in the cycle, the TNC retracts the tool by the set-up clearance, positions the tool back at rapid traverse and approaches the contour again with the defined feed rate. With machining direction Q507=0, infeed is on both sides.
6 The tool recesses to the next plunging depth.
7 The TNC repeats this process (2 to 4) until the slot depth is reached.
8 The TNC returns the tool to set-up clearance and machines a recessing traverse on both side walls.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side walls of the slot at the defined feed rate \( Q_{505} \).
3. The TNC finishes the slot floor at the defined feed rate.
4. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

- Program a positioning block to the starting position with radius compensation \( R_0 \) before the cycle call. The tool position at cycle call defines the size of the area to be machined (cycle starting point).
- Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.
- When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
- From the second infeed, the TNC reduces each further cutting traverse by 0.1 mm. This reduces lateral pressure on the tool. If the offset width \( Q_{508} \) was input into the cycle, the TNC reduces the cutting traverse by this value. After clearance roughing, the remaining material is removed with a single cut. The TNC generates an error message if the lateral offset exceeds 80 % of the effective cutting width (effective cutting width = cutting width – 2 * cutting radius).
Cycles: Turning

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

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Feed rate for plunging** Q488: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, the feed rate defined for turning is effective.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
AXIAL RECESSING 12.21
(Cycle 850, DIN/ISO: G850)

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (Q480/Q482)

- **Limit value for diameter** Q480: X value for contour limitation (diameter value)

- **Limit value Z** Q482: Z value for contour limitation

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Machining direction** Q507: Cutting direction:
  0: bidirectional (in both directions)
  1: unidirectional (in contour direction)

- **Offset width** Q508: Reduction of cutting length. After clearance roughing, the remaining material is removed with a single cut. If required, the TNC limits the programmed offset width.

- **Turning depth compensation** Q509: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor.

- **Reverse contour** Q499: Machining direction:
  0: Machining in contour direction
  1: Machining opposite the contour direction

---

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 CYCL DEF 14.0 CONTOUR</td>
</tr>
<tr>
<td>10 CYCL DEF 14.1 CONTOUR LABEL2</td>
</tr>
<tr>
<td>11 CYCL DEF 850 RECESS TURNG. AXIAL</td>
</tr>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q479=+0 ;CUTTING LIMIT</td>
</tr>
<tr>
<td>Q480=+0 ;LIMIT VALUE FOR DIAMETER</td>
</tr>
<tr>
<td>Q482=+0 ;LIMIT VALUE IN Z</td>
</tr>
<tr>
<td>Q463=+2 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q507=+0 ;MACHINING DIRECTION</td>
</tr>
<tr>
<td>Q508=+0 ;OFFSET WIDTH</td>
</tr>
<tr>
<td>Q509=+0 ;DEPTH COMPENSATION</td>
</tr>
<tr>
<td>Q499=+0 ;REVERSE CONTOUR</td>
</tr>
<tr>
<td>12 L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
<tr>
<td>14 M30</td>
</tr>
<tr>
<td>15 LBL 2</td>
</tr>
<tr>
<td>16 L X+60 Z+0</td>
</tr>
<tr>
<td>17 L Z-10</td>
</tr>
<tr>
<td>18 RND R5</td>
</tr>
<tr>
<td>19 L X+40 Z-15</td>
</tr>
<tr>
<td>20 L Z+0</td>
</tr>
<tr>
<td>21 LBL 0</td>
</tr>
</tbody>
</table>
12.22 RADIAL RECESSING
(Cycle 861, DIN/ISO: G861)

Application
This cycle enables you to radially cut in right-angled slots.
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.
The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run
The cycle processes only the area from the cycle starting point to the end point defined in the cycle.
1. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2. The TNC cuts the area between the starting position and the end point in axial direction at the defined feed rate Q478.
3. The TNC positions the tool back at rapid traverse to the beginning of cut.
4. The TNC repeats this process (1 to 3) until the slot width is reached.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.
Finishing cycle run
1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate Q505.
3 The TNC finishes half the slot width at the defined feed rate.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot side.
6 The TNC finishes the side wall of the slot at the defined feed rate Q505.
7 The TNC finishes half the slot width at the defined feed rate.
8 The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation R0 before the cycle call. The tool position at cycle call defines the size of the area to be machined (cycle starting point).
Cycles: Turning
12.22 RADIAL RECESSING
(Cycle 861, DIN/ISO: G861)

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Limit to depth** Q463: Max. recess depth per cut

NC blocks

<table>
<thead>
<tr>
<th></th>
<th>CYCL DEF 861 RADIAL RECESSING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215</td>
<td>+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460</td>
<td>+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q493</td>
<td>+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494</td>
<td>-50 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q478</td>
<td>+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483</td>
<td>+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484</td>
<td>+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505</td>
<td>+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q463</td>
<td>+0 ;LIMIT TO DEPTH</td>
</tr>
<tr>
<td></td>
<td>12 L X+75 Y+0 Z-25 FMAX M303</td>
</tr>
<tr>
<td></td>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.23  RADIAL RECESSING EXTENDED  
(Cycle 862, DIN/ISO: G862)  

Application  
This cycle enables you to radially cut in slots. Expanded scope of function:  
- You can insert a chamfer or curve at the contour start and contour end.  
- In the cycle you can define angles for the side walls of the slot  
- You can insert radii in the contour edges  
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.  
The cycle can be used for inside and outside machining. If the start diameter Q491 is larger than the end diameter Q493, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.  

Roughing cycle run  
1  The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).  
2  The TNC cuts the area between the starting position and the end point in axial direction at the defined feed rate Q478.  
3  The TNC positions the tool back at rapid traverse to the beginning of cut.  
4  The TNC repeats this process (1 to 3) until the slot width is reached.  
5  The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning
12.23 RADIAL RECESSING EXTENDED
(Cycle 862, DIN/ISO: G862)

Finishing cycle run
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate $Q_{505}$.
3. The TNC finishes half the slot width at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate $Q_{505}$.
7. The TNC finishes half the slot width at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation $R_0$ before the cycle call. The tool position at cycle call defines the size of the area to be machined (cycle starting point).
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the contour starting point

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle between the side at the contour starting point and the perpendicular to the rotary axis
### Cycles: Turning

#### 12.23 RADIAL RECESSING EXTENDED

(Cycle 862, DIN/ISO: G862)

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of starting element** Q502: Size of the starting element (chamfer section)

- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.

- **Angle of second side** Q496: Angle between the side at the contour end point and the perpendicular to the rotary axis

- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of end element** Q504: Size of the end element (chamfer section)

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Limit to depth** Q463: Max. recess depth per cut

---

### NC blocks

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>CYCL DEF 862 RADIAL RECESSING EXTENDED</td>
<td></td>
</tr>
<tr>
<td>Q215</td>
<td>+0</td>
<td>;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460</td>
<td>+2</td>
<td>;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491</td>
<td>+75</td>
<td>;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492</td>
<td>-20</td>
<td>;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493</td>
<td>+50</td>
<td>;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494</td>
<td>-50</td>
<td>;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495</td>
<td>+5</td>
<td>;ANGLE OF SIDE</td>
</tr>
<tr>
<td>Q501</td>
<td>+1</td>
<td>;TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502</td>
<td>+0.5</td>
<td>;SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q500</td>
<td>+1.5</td>
<td>;RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>Q496</td>
<td>+5</td>
<td>;ANGLE OF SECOND SIDE</td>
</tr>
<tr>
<td>Q503</td>
<td>+1</td>
<td>;TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>Q504</td>
<td>+0.5</td>
<td>;SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>Q478</td>
<td>+0.3</td>
<td>;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483</td>
<td>+0.4</td>
<td>;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484</td>
<td>+0.2</td>
<td>;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505</td>
<td>+0.2</td>
<td>;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q463</td>
<td>+0</td>
<td>;LIMIT TO DEPTH</td>
</tr>
<tr>
<td>12</td>
<td>L+75 X+0 Y+2 Z MAX M303</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>CYCL CALL</td>
<td></td>
</tr>
</tbody>
</table>
12.24 RECESSING CONTOUR RADIAL (Cycle 860, DIN/ISO: G860)

Application

This cycle enables you to radially cut in slots of any form. You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing. The cycle can be used for inside and outside machining. If the starting point of the contour is larger than the end point of the contour, the cycle runs outside machining. If the contour starting point is less than the end point, the cycle runs inside machining.

Roughing cycle run

1. The TNC positions the tool at rapid traverse in the Z coordinate (first cut-in position).
2. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
3. The TNC cuts the area between the starting position and the end point in radial direction at the defined feed rate Q478.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (2 to 4) until the slot form is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

**Finishing cycle run**

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
3. The TNC finishes one half of the slot at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate **Q505**.
7. The TNC finishes the other half of the slot at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

**Please note while programming:**

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

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

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

Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

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

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
Cycles: Turning
12.24 RECESSING CONTOUR RADIAL
(Cycle 860, DIN/ISO: G860)

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (Q480/Q482)
- **Limit value for diameter** Q480: X value for contour limitation (diameter value)
- **Limit value Z** Q482: Z value for contour limitation
- **Limit to depth** Q463: Max. recess depth per cut

### NC blocks

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>CYCL DEF 14.0 CONTOUR</td>
</tr>
<tr>
<td>10</td>
<td>CYCL DEF 14.1 CONTOUR LABEL2</td>
</tr>
<tr>
<td>11</td>
<td>CYCL DEF 860 RECESSING CONTOUR RADIAL</td>
</tr>
<tr>
<td>12</td>
<td>L X+75 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13</td>
<td>CYCL CALL</td>
</tr>
<tr>
<td>14</td>
<td>M30</td>
</tr>
<tr>
<td>15</td>
<td>LBL 2</td>
</tr>
<tr>
<td>16</td>
<td>L X+60 Z-20</td>
</tr>
<tr>
<td>17</td>
<td>L X+45</td>
</tr>
<tr>
<td>18</td>
<td>RND R2</td>
</tr>
<tr>
<td>19</td>
<td>L X+40 Z-25</td>
</tr>
<tr>
<td>20</td>
<td>L Z+0</td>
</tr>
<tr>
<td>21</td>
<td>LBL 0</td>
</tr>
</tbody>
</table>
12.25 AXIAL RECESSING
(Cycle 871, DIN/ISO: G871)

Application
This cycle enables you to axially cut in right-angled slots (face recessing).
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. The cycle processes only the area from the cycle starting point to the end point defined in the cycle.
1 The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2 The TNC cuts the area between the starting position and the end point in radial direction at the defined feed rate $Q_{478}$.
3 The TNC positions the tool back at rapid traverse to the beginning of cut.
4 The TNC repeats this process (1 to 3) until the slot width is reached.
5 The TNC positions the tool back at rapid traverse to the cycle starting point.

Finishing cycle run
1 The TNC positions the tool at rapid traverse to the first slot side.
2 The TNC finishes the side wall of the slot at the defined feed rate $Q_{505}$.
3 The TNC finishes half the slot width at the defined feed rate.
4 The TNC returns the tool at rapid traverse.
5 The TNC positions the tool at rapid traverse to the second slot side.
6 The TNC finishes the side wall of the slot at the defined feed rate $Q_{505}$.
7 The TNC finishes half the slot width at the defined feed rate.
8 The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

Please note while programming:

Program a positioning block to the starting position with radius compensation R0 before the cycle call.
The tool position at cycle call defines the size of the area to be machined (cycle starting point).

Cycle parameters

- **Machining operation** Q215: Define machining operation:
  - 0: Roughing and finishing
  - 1: Only roughing
  - 2: Only finishing to finished dimension
  - 3: Only finishing to oversize
- **Set-up clearance** Q460: Reserved, currently without function
- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)
- **Contour end in Z** Q494: Z coordinate of the contour end point
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Limit to depth** Q463: Max. recess depth per cut

NC blocks

<table>
<thead>
<tr>
<th>NC block</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 871 AXIAL RECESSING</td>
</tr>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q493=+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=-10 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q478=+0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q484=+0.2 ;OVERSIZE IN Z</td>
</tr>
<tr>
<td>Q505=+0.2 ;FINISHING FEED RATE</td>
</tr>
<tr>
<td>Q463=+0 ;LIMIT TO DEPTH</td>
</tr>
<tr>
<td>12 L X+65 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.26 AXIAL RECESSING EXTENDED  
(Cycle 872, DIN/ISO: G872)

Application
This cycle enables you to axially cut in slots (face recessing). 
Expanded scope of function:
- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than Q492 CONTOUR START IN Z, the TNC positions the tool in the Z coordinate to Q492 and begins the cycle there.
1. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
2. The TNC cuts the area between the starting position and the end point in radial direction at the defined feed rate Q478.
3. The TNC positions the tool back at rapid traverse to the beginning of cut.
4. The TNC repeats this process (1 to 3) until the slot width is reached.
5. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning
12.26 AXIAL RECESSING EXTENDED
(Cycle 872, DIN/ISO: G872)

Finishing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than Q492 CONTOUR START IN Z, the TNC positions the tool in the Z coordinate to Q492 and begins the cycle there.

1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate Q505.
3. The TNC returns the tool at rapid traverse.
4. The TNC positions the tool at rapid traverse to the second slot side.
5. The TNC finishes the side wall of the slot at the defined feed rate Q505.
6. The TNC finishes one half of the slot at the defined feed rate.
7. The TNC positions the tool at rapid traverse to the first side.
8. The TNC finishes the other half of the slot at the defined feed rate.
9. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation R0 before the cycle call. The tool position at cycle call defines the size of the area to be machined (cycle starting point).
**Cycle parameters**

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the contour starting point

- **Diameter at end of contour** Q493: X coordinate of the contour end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the contour end point

- **Angle of side** Q495: Angle between the side at the contour starting point and the parallel line to the rotary axis

- **Type of starting element** Q501: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Size of starting element** Q502: Size of the starting element (chamfer section)

- **Radius of contour edge** Q500: Radius of the inside contour edge. If no radius is specified, the radius of the cutting insert is generated.

- **Angle of second side** Q496: Angle between the side at the contour end point and the parallel line to the rotary axis

- **Type of end element** Q503: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

---

**NC blocks**

<table>
<thead>
<tr>
<th>11 CYCL DEF 871 AXIAL RECESSING EXTENDED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=+0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
</tr>
<tr>
<td>Q491=+75 ;DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>Q492=-20 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q493=+50 ;DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>Q494=-50 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q495=+5 ;ANGLE OF SIDE</td>
</tr>
<tr>
<td>Q501=+1 ;TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>Q502=+0.5 ;SIZE OF STARTING ELEMENT</td>
</tr>
</tbody>
</table>
Cycles: Turning
12.26 AXIAL RECESSING EXTENDED
(Cycle 872, DIN/ISO: G872)

- **Size of end element** Q504: Size of the end element (chamfer section)
- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour
- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction
- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.
- **Limit to depth** Q463: Max. recess depth per cut

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q500</td>
<td>+1.5</td>
</tr>
<tr>
<td>Q496</td>
<td>+5</td>
</tr>
<tr>
<td>Q503</td>
<td>+1</td>
</tr>
<tr>
<td>Q504</td>
<td>+0.5</td>
</tr>
<tr>
<td>Q478</td>
<td>+0.3</td>
</tr>
<tr>
<td>Q483</td>
<td>+0.4</td>
</tr>
<tr>
<td>Q484</td>
<td>+0.2</td>
</tr>
<tr>
<td>Q505</td>
<td>+0.2</td>
</tr>
<tr>
<td>Q463</td>
<td>+0</td>
</tr>
<tr>
<td>L</td>
<td>12</td>
</tr>
<tr>
<td>X+75</td>
<td>Y+0</td>
</tr>
<tr>
<td>Z+2</td>
<td>FMAX M303</td>
</tr>
</tbody>
</table>

13 CYCL CALL
12.27 AXIAL RECESSING
(Cycle 870, DIN/ISO: G870)

Application
This cycle enables you to axially cut in slots of any form (face recessing).
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the TNC positions the tool in the Z coordinate to the contour starting point and begins the cycle there.
1. The TNC positions the tool at rapid traverse in the X coordinate (first cut-in position).
2. The TNC runs a paraxial infeed motion at rapid traverse (lateral infeed = 0.8 cutting width).
3. The TNC cuts the area between the starting position and the end point in axial direction at the defined feed rate Q478.4
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC repeats this process (2 to 4) until the slot form is completed.
6. The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

Finishing cycle run
The TNC uses the tool position as cycle starting point when a cycle is called.
1. The TNC positions the tool at rapid traverse to the first slot side.
2. The TNC finishes the side wall of the slot at the defined feed rate \( Q_{505} \).
3. The TNC finishes one half of the slot at the defined feed rate.
4. The TNC returns the tool at rapid traverse.
5. The TNC positions the tool at rapid traverse to the second slot side.
6. The TNC finishes the side wall of the slot at the defined feed rate \( Q_{505} \).
7. The TNC finishes the other half of the slot at the defined feed rate.
8. The TNC positions the tool back at rapid traverse to the cycle starting point.

Please note while programming:

! The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits.

The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Program a positioning block to the starting position with radius compensation \( R_0 \) before the cycle call.
The tool position at cycle call defines the size of the area to be machined (cycle starting point).
Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.
When you use local \( Q_L \) parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.
### AXIAL RECESSING 12.27
(Cycle 870, DIN/ISO: G870)

#### Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Set-up clearance** Q460: Reserved, currently without function

- **Roughing feed rate** Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Oversize in diameter** Q483 (incremental): Diameter oversize for the defined contour

- **Oversize in Z** Q484 (incremental): Oversize for the defined contour in axial direction

- **Finishing feed rate** Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

- **Cutting limit** Q479: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (Q480/Q482)

- **Limit value for diameter** Q480: X value for contour limitation (diameter value)

- **Limit value Z** Q482: Z value for contour limitation

- **Limit to depth** Q463: Max. recess depth per cut

#### NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>CYCL</td>
<td>DEF 14.0 CONTOUR</td>
</tr>
<tr>
<td>10</td>
<td>CYCL</td>
<td>DEF 14.1 CONTOUR LABEL2</td>
</tr>
<tr>
<td>11</td>
<td>CYCL</td>
<td>DEF 870 AXIAL RECESSING</td>
</tr>
<tr>
<td></td>
<td>Q215=+0</td>
<td>;MACHINING OPERATION</td>
</tr>
<tr>
<td></td>
<td>Q460=+2</td>
<td>;SAFETY CLEARANCE</td>
</tr>
<tr>
<td></td>
<td>Q478=+0.3</td>
<td>;ROUGHING FEED RATE</td>
</tr>
<tr>
<td></td>
<td>Q483=+0.4</td>
<td>;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td></td>
<td>Q484=+0.2</td>
<td>;OVERSIZE IN Z</td>
</tr>
<tr>
<td></td>
<td>Q505=+0.2</td>
<td>;FINISHING FEED RATE</td>
</tr>
<tr>
<td></td>
<td>Q479=+0</td>
<td>;CUTTING LIMIT</td>
</tr>
<tr>
<td></td>
<td>Q480=+0</td>
<td>;LIMIT VALUE FOR DIAMETER</td>
</tr>
<tr>
<td></td>
<td>Q482=+0</td>
<td>;LIMIT VALUE IN Z</td>
</tr>
<tr>
<td></td>
<td>Q463=+0</td>
<td>;LIMIT TO DEPTH</td>
</tr>
<tr>
<td>12</td>
<td>L X+75 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>CYCL</td>
<td>CALL</td>
</tr>
<tr>
<td>14</td>
<td>M30</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>LBL</td>
<td>2</td>
</tr>
<tr>
<td>16</td>
<td>L X+60 Z+0</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>L Z-10</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>RND</td>
<td>R5</td>
</tr>
<tr>
<td>19</td>
<td>L X+40 Z-15</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>L Z+0</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>LBL</td>
<td>0</td>
</tr>
</tbody>
</table>

HEIDENHAIN | TNC 640 | User’s manual for cycle programming | 9/2015

423
Cycles: Turning

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

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

Application
This cycle enables you to run longitudinal turning of threads. You can process single threads or multi-threads with the cycle. If you do not enter a thread depth, the cycle uses thread depth in accordance with the ISO1502 standard. The cycle can be used for inside and outside machining.

Cycle run
The TNC uses the tool position as cycle starting point when a cycle is called.
1. The TNC positions the tool in rapid traverse at set-up clearance in front of the thread and runs an infeed motion.
2. The TNC runs a paraxial longitudinal cut. Here the TNC synchronizes feed rate and speed so that the defined pitch is machined.
3. The TNC retracts the tool at rapid traverse by the set-up clearance.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC runs an infeed motion. The infeeds are run according to the angle of infeed \(Q467\).
6. The TNC repeats the process (2 to 5) until the thread depth is completed.
7. The TNC runs the number of air cuts as defined in \(Q476\).
8. The TNC repeats the process (2 to 7) according to the number of traverses \(Q475\).
9. The TNC positions the tool back at rapid traverse to the cycle starting point.
Please note while programming:

Program a positioning block to the starting position with radius compensation R0 before the cycle call. The TNC uses the set-up clearance Q460 as approach path. The approach path must be long enough for the feed axes to be accelerated to the required velocity.

The TNC uses the thread pitch as overrun path. The overrun path must be long enough to decelerate the feed axes.

Parameters are available for approach and overrun in Cycle 832 THREAD EXTENDED.

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

With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use an external tool for inside machining, you can execute machining in the negative diameter range (-X) and reverse the direction of workpiece rotation. Note that with pre-positioning in the negative diameter range, the TNC reverses the effect of the parameter G471 Thread position (external thread is then 1 and internal thread 0).

The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.
Cycles: Turning

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

Cycle parameters

- **Thread position** Q471: Define the position of the thread:
  - 0: External thread
  - 1: Internal thread

- **Set-up clearance** Q460: Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) to the synchronized feed rate.

- **Thread diameter** Q491: Define the nominal diameter of the thread.

- **Thread pitch** Q472: Pitch of the thread.

- **Depth of thread** Q473 (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.

- **Contour start in Z** Q492: Z coordinate of the starting point

- **Contour end in Z** Q494: Z coordinate of the end point including the runout of the thread Q474.

- **Runout of thread** Q474 (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.

- **Maximum cutting depth** Q463: Maximum plunging depth in radial direction relative to the radius.

- **Angle of infeed** Q467: Angle for the infeed Q463. The reference angle is formed by the perpendicular to the rotary axis.

- **Type of infeed** Q468: Define the type of infeed:
  - 0: Constant chip cross section (infeed lessens with depth)
  - 1: Constant plunging depth

- **Starting angle** Q470: Angle of the turning spindle at which the thread start is to be made.

- **Number of starts** Q475: Number of thread starts

- **Number of air cuts** Q476: Number of air cuts without infeed at finished thread depth

NC blocks

<table>
<thead>
<tr>
<th>11 CYCL DEF 831 LONGITUDINAL THREAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q471=+0 ;THREAD POSITION</td>
</tr>
<tr>
<td>Q460=+5 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q491=+75 ;THREAD DIAMETER</td>
</tr>
<tr>
<td>Q472=+2 ;THREAD PITCH</td>
</tr>
<tr>
<td>Q473=+0 ;THREAD DEPTH</td>
</tr>
<tr>
<td>Q492=+0 ;CONTOUR START IN Z</td>
</tr>
<tr>
<td>Q494=−15 ;CONTOUR END IN Z</td>
</tr>
<tr>
<td>Q474=+0 ;RUNOUT OF THREAD</td>
</tr>
<tr>
<td>Q463=+0.5 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q467=+30 ;ANGLE OF INFEED</td>
</tr>
<tr>
<td>Q468=+0 ;TYPE OF INFEED</td>
</tr>
<tr>
<td>Q470=+0 ;START ANGLE</td>
</tr>
<tr>
<td>Q475=+30 ;NUMBER OF STARTS</td>
</tr>
<tr>
<td>Q476=+30 ;NO. OF AIR CUTS</td>
</tr>
<tr>
<td>12 L X+80 Y+0 Z+2 FMAX M303</td>
</tr>
<tr>
<td>13 CYCL CALL</td>
</tr>
</tbody>
</table>
12.29 THREAD EXTENDED (Cycle 832, DIN/ISO: G832)

Application

This cycle enables you to run both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- Selection of longitudinal thread or face thread.
- The parameters for dimension type of taper, taper angle and contour starting point X enable the definition of various tapered threads.
- The parameters for approach path and overrun path define a path in which feed axes can be accelerated or decelerated.

You can process single threads or multi-threads with the cycle. If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle run

The TNC uses the tool position as cycle starting point when a cycle is called.

1. The TNC positions the tool in rapid traverse at set-up clearance in front of the thread and runs an infeed motion.
2. The TNC runs a longitudinal cut. Here the TNC synchronizes feed rate and speed so that the defined pitch is machined.
3. The TNC retracts the tool at rapid traverse by the set-up clearance.
4. The TNC positions the tool back at rapid traverse to the beginning of cut.
5. The TNC runs an infeed motion. The infeeds are run according to the angle of infeed Q467.
6. The TNC repeats the process (2 to 5) until the thread depth is completed.
7. The TNC runs the number of air cuts as defined in Q476.
8. The TNC repeats the process (2 to 7) according to the number of traverses Q475.
9. The TNC positions the tool back at rapid traverse to the cycle starting point.
Please note while programming:

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

The approach path (Q465) must be long enough for the feed axes to be accelerated to the required velocity.

The overrun path (Q466) must be long enough to decelerate the feed axes.

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

With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use an external tool for inside machining, you can execute machining in the negative diameter range (-X) and reverse the direction of workpiece rotation. Note that with pre-positioning in the negative diameter range, the TNC reverses the effect of the parameter G471 Thread position (external thread is then 1 and internal thread 0).

The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.
Cycle parameters

- **Thread position** Q471: Define the position of the thread:
  0: External thread
  1: Internal thread

- **Thread orientation** Q461: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)

- **Set-up clearance** Q460: Set-up clearance perpendicular to thread pitch.

- **Thread pitch** Q472: Pitch of the thread.

- **Depth of thread** Q473 (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.

- **Dimension type of taper** Q464: Define the type of dimension for the taper contour:
  0: Via starting point and end point
  1: Via end point, start-X and taper angle
  2: Via end point, start-Z and taper angle
  3: Via starting point, end-X and taper angle
  4: Via starting point, end-Z and taper angle

- **Diameter at contour start** Q491: X coordinate of the contour starting point (diameter value)

- **Contour start in Z** Q492: Z coordinate of the starting point

- **Diameter at end of contour** Q493: X coordinate of the end point (diameter value)

- **Contour end in Z** Q494: Z coordinate of the end point

- **Taper angle** Q469: Taper angle of contour

- **Runout of thread** Q474 (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.

- **Approach path** Q465 (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside of the defined thread contour.

- **Overrun path** Q466: Length of the path in pitch direction on which the feed axes are decelerated. The overrun path is within the defined thread contour.

- **Maximum cutting depth** Q463: Maximum plunging depth perpendicular to the thread pitch

---

NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 CYCL DEF 832 THREAD EXTENDED</td>
<td></td>
</tr>
<tr>
<td>Q471=+0 ;THREAD POSITION</td>
<td></td>
</tr>
<tr>
<td>Q461=+0 ;THREAD ORIENTATION</td>
<td></td>
</tr>
<tr>
<td>Q460=+2 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q472=+2 ;THREAD PITCH</td>
<td></td>
</tr>
<tr>
<td>Q473=+0 ;THREAD DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q464=+0 ;DIMENSION TYPE FOR TAPER</td>
<td></td>
</tr>
<tr>
<td>Q491=+100 ;DIAMETER AT CONTOUR START</td>
<td></td>
</tr>
<tr>
<td>Q492=+0 ;CONTOUR START IN Z</td>
<td></td>
</tr>
<tr>
<td>Q493=+110 ;DIAMETER AT END OF CONTOUR</td>
<td></td>
</tr>
<tr>
<td>Q494=+35 ;CONTOUR END IN Z</td>
<td></td>
</tr>
<tr>
<td>Q469=+0 ;TAPER ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q474=+0 ;RUNOUT OF THREAD</td>
<td></td>
</tr>
<tr>
<td>Q465=+4 ;APPROACH PATH</td>
<td></td>
</tr>
<tr>
<td>Q466=+4 ;OVERRUN PATH</td>
<td></td>
</tr>
<tr>
<td>Q463=+0.5 ;MAX. CUTTING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q467=+30 ;ANGLE OF INFEED</td>
<td></td>
</tr>
<tr>
<td>Q468=+0 ;TYPE OF INFEED</td>
<td></td>
</tr>
<tr>
<td>Q470=+0 ;START ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q475=+30 ;NUMBER OF STARTS</td>
<td></td>
</tr>
<tr>
<td>Q476=+30 ;NO. OF AIR CUTS</td>
<td></td>
</tr>
<tr>
<td>12 L X+80 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL</td>
<td></td>
</tr>
</tbody>
</table>
12.29 THREAD EXTENDED (Cycle 832, DIN/ISO: G832)

- **Angle of infeed** Q467: Angle for the infeed Q463. The reference angle is formed by the parallel line to the thread pitch.
- **Type of infeed** Q468: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Starting angle** Q470: Angle of the turning spindle at which the thread start is to be made.
- **Number of starts** Q475: Number of thread starts
- **Number of air cuts** Q476: Number of air cuts without infeed at finished thread depth
12.30 CONTOUR-PARALLEL THREAD
(Cycle 830, DIN/ISO: G830)

Application
This cycle enables you to run both face turning and longitudinal turning of threads with any form.
You can process single threads or multi-threads with the cycle.
If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.
The cycle can be used for inside and outside machining.

! The cycle 830 runs the overrun Q466 following the programmed contour. Note the spatial conditions.

Cycle run
The TNC uses the tool position as cycle starting point when a cycle is called.
1 The TNC positions the tool in rapid traverse at set-up clearance in front of the thread and runs an infeed motion.
2 The TNC runs a thread cut parallel to the defined thread contour. Here the TNC synchronizes feed rate and speed so that the defined pitch is machined.
3 The TNC retracts the tool at rapid traverse by the set-up clearance.
4 The TNC positions the tool back at rapid traverse to the beginning of cut.
5 The TNC runs an infeed motion. The infeeds are run according to the angle of infeed Q467.
6 The TNC repeats the process (2 to 5) until the thread depth is completed.
7 The TNC runs the number of air cuts as defined in Q476.
8 The TNC repeats the process (2 to 7) according to the number of traverses Q475.
9 The TNC positions the tool back at rapid traverse to the cycle starting point.
Cycles: Turning

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

Please note while programming:

- Program a positioning block to the starting position with radius compensation R0 before the cycle call.
- The approach path (Q465) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (Q466) must be long enough to decelerate the feed axes.
- Both the approach and overrun take place outside the defined contour.
- When the TNC runs a thread cut, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).
- Before calling the cycle you must program the cycle 14 CONTOUR to define the subprogram number.
- When you use local QL Q parameters in a contour subprogram you must also assign or calculate these in the contour subprogram.

With some machine types the turning tool is not clamped in the milling spindle but in a separate holder adjacent to the spindle. The turning tool cannot be rotated through 180° in such cases to machine internal and external threads with only one tool for example. If with such a machine you wish to use an external tool for inside machining, you can execute machining in the negative diameter range (-X) and reverse the direction of workpiece rotation.
- Note that with pre-positioning in the negative diameter range, the TNC reverses the effect of the parameter G471 Thread position (external thread is then 1 and internal thread 0).
- The retraction motion is directly to the starting position. Always position the tool so that the TNC can approach the starting point at the end of the cycle without collisions.
**Cycle parameters**

- **Thread position** Q471: Define the position of the thread:
  - 0: External thread
  - 1: Internal thread

- **Thread orientation** Q461: Define the direction of the thread pitch:
  - 0: Longitudinal (parallel to the rotary axis)
  - 1: Lateral (perpendicular to the rotary axis)

- **Set-up clearance** Q460: Set-up clearance perpendicular to thread pitch.

- **Thread pitch** Q472: Pitch of the thread.

- **Depth of thread** Q473 (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.

- **Runout of thread** Q474 (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.
### Cycles: Turning

#### 12.30 CONTOUR-PARALLEL THREAD

(Cycle 830, DIN/ISO: G830)

- **Approach path** Q465 (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside of the defined thread contour.

- **Overrun path** Q466: Length of the path in pitch direction on which the feed axes are decelerated. The overrun path is within the defined thread contour.

- **Maximum cutting depth** Q463: Maximum plunging depth perpendicular to the thread pitch.

- **Angle of infeed** Q467: Angle for the infeed Q463. The reference angle is formed by the parallel line to the thread pitch.

- **Type of infeed** Q468: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth

- **Starting angle** Q470: Angle of the turning spindle at which the thread start is to be made.

- **Number of starts** Q475: Number of thread starts.

- **Number of air cuts** Q476: Number of air cuts without infeed at finished thread depth.

---

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 CYCL DEF 14.0 CONTOUR</td>
<td></td>
</tr>
<tr>
<td>10 CYCL DEF 14.1 CONTOUR LABEL2</td>
<td></td>
</tr>
<tr>
<td>11 CYCL DEF 830 CONTOUR-PARALLEL THREAD</td>
<td></td>
</tr>
<tr>
<td>Q471+0 : THREAD POSITION</td>
<td></td>
</tr>
<tr>
<td>Q461+0 : THREAD ORIENTATION</td>
<td></td>
</tr>
<tr>
<td>Q460+2 : SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td>Q472+2 : THREAD PITCH</td>
<td></td>
</tr>
<tr>
<td>Q473+0 : THREAD DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q474+0 : RUNOUT OF THREAD</td>
<td></td>
</tr>
<tr>
<td>Q465+4 : APPROACH PATH</td>
<td></td>
</tr>
<tr>
<td>Q466+4 : OVERRUN PATH</td>
<td></td>
</tr>
<tr>
<td>Q463+0.5 : MAX. CUTTING DEPTH</td>
<td></td>
</tr>
<tr>
<td>Q467+30 : ANGLE OF INFEED</td>
<td></td>
</tr>
<tr>
<td>Q468+0 : TYPE OF INFEED</td>
<td></td>
</tr>
<tr>
<td>Q470+0 : START ANGLE</td>
<td></td>
</tr>
<tr>
<td>Q475+30 : NUMBER OF STARTS</td>
<td></td>
</tr>
<tr>
<td>Q476+30 : NO. OF AIR CUTS</td>
<td></td>
</tr>
<tr>
<td>12 L X+80 Y+0 Z+2 FMAX M303</td>
<td></td>
</tr>
<tr>
<td>13 CYCL CALL</td>
<td></td>
</tr>
<tr>
<td>14 M30</td>
<td></td>
</tr>
<tr>
<td>15 LBL 2</td>
<td></td>
</tr>
<tr>
<td>16 L X+60 Z+0</td>
<td></td>
</tr>
<tr>
<td>17 L X+70 Z-30</td>
<td></td>
</tr>
<tr>
<td>18 RND R60</td>
<td></td>
</tr>
<tr>
<td>19 L Z-45</td>
<td></td>
</tr>
<tr>
<td>20 LBL 0</td>
<td></td>
</tr>
</tbody>
</table>
12.31 GEAR HOBBING (Cycle 880, DIN/ISO: G880)

Cycle run

With Cycle 880 Gear Hobbing you can machine external cylindrical gears or helical gears with any angles. In the cycle you first define the gear and then the tool with which the gear is to be machined. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary motion of the tool spindle and rotary table. In addition, the gear hob moves along the workpiece in axial direction.

While Cycle 880 Gear Hobbing is active, the coordinate system might be rotated. It is therefore essential to program Cycle 801 RESET ROTARY COORDINATE SYSTEM and M145 at the end of the cycle.

Cycle run:

1. The TNC positions the tool in the tool axis to clearance height Q260 at the feed rate FMAX. If the value of the current tool location in the tool axis is greater than Q260, the tool is not moved.

2. Before tilting the working plane, the TNC positions the tool in X to a safe coordinate at the feed rate FMAX. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.

3. The TNC then tilts the working plane at the feed rate Q253; M144 is internally active in the cycle.

4. The TNC positions the tool at the feed rate FMAX to the starting point in the working plane.

5. The TNC then moves the tool in the tool axis at the feed rate Q253 to the set-up clearance Q460.

6. The TNC moves the hob in the longitudinal direction at the programmed feed rate Q478 (for roughing) or Q505 (for finishing) along the workpiece into which the teeth are to be cut. The area to be machined is limited by the starting point in Z Q551+Q460 and the end point in Z Q552+Q460.

7. When the tool reaches the end point, it is retracted at the feed rate Q253 and returns to the starting point.

8. The TNC repeats the steps 5 to 7 until the defined gear is completed.

9. Finally, the TNC retracts the tool to the clearance height Q260 at the feed rate FMAX.

10. The machining operation ends in the tilted system.

11. Now you need to move the tool to a safe height and reset the tilting of the working plane.

12. Then you must program Cycle 801 RESET ROTARY COORDINATE SYSTEM and M145.
Cycles: Turning
12.31 GEAR HOBBING (Cycle 880, DIN/ISO: G880)

Please note while programming:

- The values entered for module, number of teeth and outside diameter are monitored. If these values are not consistent, an error message is displayed. It is also possible to make entries for only 2 of these 3 parameters. In this case, enter the value 0 for either the module or the number of teeth or the outside diameter. The TNC then calculates the missing value. Program FUNCTION TURNDATA SPIN VCONST:OFF.

- If you program FUNCTION TURNDATA SPIN VCONST:OFF S15, the spindle speed of the tool is calculated as follows: Q541 x S. With Q541=238 and S=15, this would result in a tool spindle speed of 3570 rpm.

- Define the tool as a milling cutter in the tool table.
- To avoid exceeding the maximum permissible spindle speed of the tool, you can enter a limitation. (Entry in the "Nmax" column of the tool table "tool.t".)
- Before starting the cycle, program the direction of rotation of the workpiece (M303/M304).
- Before cycle call, set the datum to the center of rotation.

Cycle 880 Gear Hobbing is run in turning mode and is CALLactive.
- Software option 50 must be enabled

Danger of collision!

- Pre-position the tool so that it is already on the desired machining side Q550. On this machining side, move the tool to a safe position where there is no danger of collision with the workpiece (clamping devices) during tilting.
- Please note that the starting point in Z and the end point in Z are extended by the set-up clearance Q460! Clamp the workpiece in such a way that there is no danger of collision between the tool and the clamping devices!

- If you program M136 before the cycle, the TNC interprets feed rate values in the cycle in mm/rev., and if you are not using M136, in mm/min!

- After Cycle 880 GEAR HOBBING remember to call Cycle 801 and M145 in order to reset the coordinate system.

- If you abort a program during machining, you must reset the coordinate system with Cycle 801 and call M145 before you start machining again!
Cycle parameters

- **Machining operation** Q215: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize

- **Module** Q540: Define the gear: Module of the gear wheel. Input range 0 to 99.9999

- **Number of teeth** Q541: Define the gear: Number of teeth. Input range 0 to 99999

- **Outside diameter** Q542: Define the gear: Outside diameter of the finished part. Input range 0 to 99999.9999

- **Trough-tip clearance** Q543: Define the gear: Distance between the tip circle of the gear to be cut and the root circle of the mating gear. Input range 0 to 9.9999

- **Angle of inclination** Q544: Define the gear: Angle by which the teeth in a helical gear are inclined relative to the axis direction. (With a straight-cut gear, this angle is 0°) Input range -45 to +45

- **Tool lead angle** Q545: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (e.g. 0°47’=0.7833) Input range -60.0000 to +60.0000

- **Change tool direction (3, 4)** Q546: Define the tool: Direction of spindle rotation of the gear hob:
  3: Right-turning tool (M3)
  4: Left-turning tool (M4)

- **Ang. offset, spindle** Q547: Angle by which the TNC rotates the workpiece at cycle start. Input range -180.0000 to +180.0000

- **Machining side** Q550: Define the side on which the machining operation is to be performed.
  0: Positive machining side
  1: Negative machining side

- **Preferred direction** Q533: Selection of alternate possibilities of inclination.
  0: Option with shortest way
  -1: Option with negative direction
  +1: Option with positive direction
  -2: Option with negative direction at a range of between -90° and -180°
  +2: Option with positive direction at a range of between +90° and +180°

---

### NC blocks

<table>
<thead>
<tr>
<th>63 CYCL DEF 880 GEAR HOBBING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q215=0 ;MACHINING OPERATION</td>
</tr>
<tr>
<td>Q540=0 ;MODULE</td>
</tr>
<tr>
<td>Q541=0 ;NUMBER OF TEETH</td>
</tr>
<tr>
<td>Q542=0 ;OUTSIDE DIAMETER</td>
</tr>
<tr>
<td>Q543=0.167;TROUGH-TIP CLEARANCE</td>
</tr>
<tr>
<td>Q544=0 ;ANGLE OF INCLINATION</td>
</tr>
<tr>
<td>Q545=0 ;TOOL LEAD ANGLE</td>
</tr>
<tr>
<td>Q546=3 ;CHANGE TOOL DIRECTN.</td>
</tr>
<tr>
<td>Q547=0 ;ANG. OFFSET, SPINDLE</td>
</tr>
<tr>
<td>Q550=1 ;MACHINING SIDE</td>
</tr>
<tr>
<td>Q553=0 ;PREFERRED DIRECTION</td>
</tr>
<tr>
<td>Q530=2 ;INCLINED MACHINING</td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q260=100 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q553=10 ;TOOL LENGTH OFFSET</td>
</tr>
<tr>
<td>Q551=0 ;STARTING POINT IN Z</td>
</tr>
<tr>
<td>Q552=-10 ;END POINT IN Z</td>
</tr>
<tr>
<td>Q463=1 ;MAX. CUTTING DEPTH</td>
</tr>
<tr>
<td>Q460=2 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q488=0.3 ;PLUNGING FEED RATE</td>
</tr>
<tr>
<td>Q478=0.3 ;ROUGHING FEED RATE</td>
</tr>
<tr>
<td>Q483=0.4 ;OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>Q505=0.2 ;FINISHING FEED RATE</td>
</tr>
</tbody>
</table>
- **Inclined machining** Q530: Position the tilting axes for inclined machining:
  1. Position the tilting axis automatically, thereby orienting the tool tip (MOVE). The relative position between the tool and workpiece remains unchanged. The TNC performs a compensating movement with the linear axes
  2. Position the tilting axis automatically without orienting the tool tip (TURN).

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Entry in mm/min. Input range 0 to 99999.9999 alternatively FMAX, FAUTO, PREDEF

- **Tool length offset** Q553: Define which section of the gear hob is to be used. As gear hobbing causes wear to the hob teeth, the tool can be offset in the longitudinal direction to evenly apply the load over the entire length of the tool. In parameter Q553 you enter an incremental distance by which the tool is to be moved in the longitudinal direction. Input range 0 to 99.9999

- **Starting point in Z** Q551: Starting point in Z for gear hobbing. Input range -99999.9999 to 99999.9999

- **End point in Z** Q552: End point in Z for gear hobbing. Input range -99999.9999 to 99999.9999

- **Maximum cutting depth** Q463: Maximum infeed (radius value) in radial direction. The infeed is divided evenly to avoid abrasive cuts. Input range 0.001 to 999.999

- **Set-up clearance** Q460 (incremental): Distance for retraction and pre-positioning. Input range 0 to 999.999

- **Feed rate for plunging** Q488: Feed rate for tool infeed. Input range 0 to 99999.999
Roughing feed rate Q478: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

Oversize in diameter Q483 (incremental): Diameter oversize for the defined contour.

Finishing feed rate Q505: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the TNC in millimeters per revolution, without M136 in millimeters per minute.

Direction of rotation depending on the machining side (Q550)

Determine the direction of rotation of the rotary table:

1. What tool? (Right-cutting / left-cutting)?
2. What machining side? X+ (Q550=0) / X- (Q550=1)
3. Look up the direction of rotation of the rotary table in one of the two tables below! Select the table for the direction of rotation of your tool (right-cutting/left-cutting). In this table, look up the direction of rotation of the rotary table for the desired machining side X+ (Q550=0) / X- (Q550=1).

<table>
<thead>
<tr>
<th>Tool: Right-cutting M3</th>
</tr>
</thead>
</table>
| Machining side X+ (Q550=0) | Direction of table rotation: clockwise (M303)  
| Machining side X- (Q550=1) | Direction of table rotation: counterclockwise (M304) |

<table>
<thead>
<tr>
<th>Tool: Left-cutting M4</th>
</tr>
</thead>
</table>
| Machining side X+ (Q550=0) | Direction of table rotation: counterclockwise (M304)  
| Machining side X- (Q550=1) | Direction of table rotation: clockwise (M303) |
12.32 CHECK UNBALANCE (Cycle 892, DIN/ISO: G892)

Application
When turning a nonsymmetrical workpiece, such as a pump body, an unbalance may occur. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle 892 CHECK UNBALANCE the TNC checks the unbalance of the turning spindle. This cycle uses two parameters. Q450 describes the maximum unbalance and Q451 the maximum speed. If the maximum unbalance is exceeded, an error message is displayed and the program is aborted. When the maximum unbalance is not exceeded, the TNC executes the program without interruptions. This function protects the machine mechanics. It enables you to take action if excessive unbalance is detected.
Please note while programming:

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance.

The removal of material during machining will change the mass distribution within the workpiece. This may also have an influence on workpiece unbalance. Therefore, unbalance checks should also be carried out between machining steps.

Keep in mind the mass and unbalance of the workpiece when choosing the speed. Do not use high speeds with heavy workpieces or high unbalance loads.

Software option 50 must be enabled

This function is executed in turning mode. FUNCTION MODE TURN must be active, otherwise the TNC generates an error message.

Your machine tool builder configures Cycle 892.

Your machine tool builder defines the function of Cycle 892.

The turning spindle rotates during the unbalance check.

This function can also be run on machines with more than one turning spindle. Contact the machine tool builder for further information.

You need to check the applicability of the control’s internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.

If Cycle 892 CHECK UNBALANCE has aborted a program, it is recommended to use the manual MEASURE UNBALANCE cycle. With this cycle, the TNC determines the unbalance and calculates the mass and position of a balancing weight. For more information on the manual MEASURE UNBALANCE cycle, refer to the User’s Manual for Conversational Programming.
Cycles: Turning

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

Cycle parameters

- **Maximum runout** Q450: (mm) Specifies the maximum amplitude of a sinusoidal unbalance signal. The signal results from the following error of the measuring axis and from the spindle revolutions.

- **Speed** Q451: (rpm) The unbalance check starts at a low rotational speed (e.g. 50 rpm). It is automatically increased by specified increments (e.g. 25 rpm) until the defined maximum speed is reached. Spindle override is not effective.

NC blocks

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>63 CYCL DEF 892 CHECK UNBALANCE</td>
<td></td>
</tr>
<tr>
<td>Q450=0 ;MAXIMUM RUNOUT</td>
<td></td>
</tr>
<tr>
<td>Q451=50 ;SPEED</td>
<td></td>
</tr>
</tbody>
</table>
### Example program 12.33

**Example: Shoulder with recess**

![Diagram of Shoulder with Recess](image)

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
<th>Code</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM SHOULDER MM</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>BLK FORM 0.1 Y X+0 Y-10 Z-35</td>
<td>Definition of workpiece blank</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>BLK FORM 0.2 X+87 Y+10 Z+2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>TOOL CALL 12</td>
<td>Tool call</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>M140 MB MAX</td>
<td>Retract the tool</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>FUNCTION MODE TURN</td>
<td>Activate Turning mode</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>FUNCTION TURNDATA SPIN VCONST:ON VC:150</td>
<td>Constant surface speed</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>CYCL DEF 800 ADAPT ROTARY COORDINATE SYSTEM</td>
<td>Cycle definition adapt rotary coordinate system</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q497=0 ;PRECISION ANGLE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q498=0 ;REVERSE TOOL</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>M136</td>
<td>Feed rate in mm per revolution</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>L X+165 Y+0 R0 FMAX</td>
<td>Move to starting point in the plane</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>L Z+2 R0 FMAX M304</td>
<td>Set-up clearance, turning spindle on</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>CYCL DEF 812 SHOULDER LONG. EXTENDED.</td>
<td>Cycle definition shoulder longitudinal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q215=0 ;MACHINING OPERATION</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q460=+2 ;SAFETY CLEARANCE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q491=+160 ;DIAMETER AT CONTOUR START</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q492=0 ;CONTOUR START IN Z</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q493=+150 ;DIAMETER AT END OF CONTOUR</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q494=-40 ;CONTOUR END IN Z</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q495=0 ;ANGLE OF CIRCUM. SURFACE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q501=+1 ;TYPE OF STARTING ELEMENT</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q502=+2 ;SIZE OF STARTING ELEMENT</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q500=+1 ;RADIUS OF CONTOUR EDGE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q496=+0 ;ANGLE OF FACE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q503=+1 ;TYPE OF END ELEMENT</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q504=+2 ;SIZE OF END ELEMENT</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q463=+2.5 ;MAX. CUTTING DEPTH</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q478=+0.25 ;ROUGHING FEED RATE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q483=+0.4 ;OVERSIZE FOR DIAMETER</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
# Cycles: Turning

## 12.33 Example program

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>483</td>
<td>Q497=+0</td>
<td>; PRECISION ANGLE</td>
</tr>
<tr>
<td>484</td>
<td>Q498=+0</td>
<td>; REVERSE TOOL</td>
</tr>
<tr>
<td>473</td>
<td>12 CYCL CALL M8</td>
<td>Cycle call</td>
</tr>
<tr>
<td>474</td>
<td>13 M305</td>
<td>Turning spindle off</td>
</tr>
<tr>
<td>475</td>
<td>14 TOOL CALL 15</td>
<td>Tool call</td>
</tr>
<tr>
<td>476</td>
<td>15 M140 MB MAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>477</td>
<td>16 FUNCTION TURNDATA SPIN VCONST:ON VC:100</td>
<td>Constant cutting speed</td>
</tr>
<tr>
<td>478</td>
<td>17 CYCL DEF 800 ADAPT ROTARY COORDINATE SYSTEM</td>
<td>Cycle definition adapt rotary coordinate system</td>
</tr>
<tr>
<td>479</td>
<td>Q497=+0</td>
<td>; PRECISION ANGLE</td>
</tr>
<tr>
<td>480</td>
<td>Q498=+0</td>
<td>; REVERSE TOOL</td>
</tr>
<tr>
<td>481</td>
<td>18 L X+165 Y+0 R0 FMAX</td>
<td>Move to starting point in the plane</td>
</tr>
<tr>
<td>482</td>
<td>19 L Z+2 R0 FMAX M304</td>
<td>Set-up clearance, turning spindle on</td>
</tr>
<tr>
<td>483</td>
<td>20 CYCL DEF 862 RADIAL RECESSING EXTENDED</td>
<td>Cycle definition recess</td>
</tr>
<tr>
<td>484</td>
<td>Q215=+0</td>
<td>; MACHINING OPERATION</td>
</tr>
<tr>
<td>485</td>
<td>Q460=+2</td>
<td>; SAFETY CLEARANCE</td>
</tr>
<tr>
<td>486</td>
<td>Q491=+150</td>
<td>; DIAMETER AT CONTOUR START</td>
</tr>
<tr>
<td>487</td>
<td>Q492=+12</td>
<td>; CONTOUR START IN Z</td>
</tr>
<tr>
<td>488</td>
<td>Q493=+142</td>
<td>; DIAMETER AT END OF CONTOUR</td>
</tr>
<tr>
<td>489</td>
<td>Q494=+18</td>
<td>; CONTOUR END IN Z</td>
</tr>
<tr>
<td>490</td>
<td>Q495=+0</td>
<td>; ANGLE OF SIDE</td>
</tr>
<tr>
<td>491</td>
<td>Q501=+1</td>
<td>; TYPE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>492</td>
<td>Q502=+1</td>
<td>; SIZE OF STARTING ELEMENT</td>
</tr>
<tr>
<td>493</td>
<td>Q500=+0</td>
<td>; RADIUS OF CONTOUR EDGE</td>
</tr>
<tr>
<td>494</td>
<td>Q496=+0</td>
<td>; ANGLE OF SECOND SIDE</td>
</tr>
<tr>
<td>495</td>
<td>Q503=+1</td>
<td>; TYPE OF END ELEMENT</td>
</tr>
<tr>
<td>496</td>
<td>Q504=+1</td>
<td>; SIZE OF END ELEMENT</td>
</tr>
<tr>
<td>497</td>
<td>Q478=+0.3</td>
<td>; ROUGHING FEED RATE</td>
</tr>
<tr>
<td>498</td>
<td>Q483=+0.4</td>
<td>; OVERSIZE FOR DIAMETER</td>
</tr>
<tr>
<td>499</td>
<td>Q484=+0.2</td>
<td>; OVERSIZE IN Z</td>
</tr>
<tr>
<td>500</td>
<td>Q505=+0.15</td>
<td>; FINISHING FEED RATE</td>
</tr>
<tr>
<td>501</td>
<td>Q463=+0</td>
<td>; LIMIT TO DEPTH</td>
</tr>
<tr>
<td>502</td>
<td>21 CYCL CALL M8</td>
<td>Cycle call</td>
</tr>
<tr>
<td>503</td>
<td>22 M305</td>
<td>Turning spindle off</td>
</tr>
<tr>
<td>504</td>
<td>23 M137</td>
<td>Feed rate in mm per minute</td>
</tr>
<tr>
<td>505</td>
<td>24 M140 MB MAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>506</td>
<td>25 FUNCTION MODE MILL</td>
<td>Activate Milling mode</td>
</tr>
<tr>
<td>507</td>
<td>26 M30</td>
<td>End of program</td>
</tr>
<tr>
<td>508</td>
<td>27 END PGM SHOULDER MM</td>
<td></td>
</tr>
</tbody>
</table>
Example: Gear hobbing

Cycle 880 GEAR HOBBING is used in the following program. This programming example illustrates the machining of a helical gear, with Module=2.1.

Program sequence
- Tool call: Gear hob
- Start turning mode
- Approach safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>BEGIN PGM 5 MM</td>
</tr>
<tr>
<td>1</td>
<td>BLK FORM CYLINDER Z R42 L150 Definition of workpiece blank: Cylinder</td>
</tr>
<tr>
<td>2</td>
<td>FUNCTION MODE MILL Activate milling mode</td>
</tr>
<tr>
<td>3</td>
<td>TOOL CALL “GEAR_HOB_D75” Call the tool</td>
</tr>
<tr>
<td>4</td>
<td>FUNCTION MODE TURN Activate turning mode</td>
</tr>
<tr>
<td>5</td>
<td>CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM Reset the coordinate system</td>
</tr>
<tr>
<td>6</td>
<td>M145 Deactivate M144 if still active</td>
</tr>
<tr>
<td>7</td>
<td>FUNCTION TURNDATA SPIN VCONST:OFF 550 Constant surface speed OFF</td>
</tr>
<tr>
<td>8</td>
<td>M140 MB MAX Retract the tool</td>
</tr>
<tr>
<td>9</td>
<td>L A+0 R0 FMAX Set the rotary axis to 0</td>
</tr>
<tr>
<td>10</td>
<td>L X+250 Y-250 R0 FMAX Pre-position the tool in the working plane on the side on which machining will be performed</td>
</tr>
<tr>
<td>11</td>
<td>Z+20 R0 FMAX Pre-position the tool in the spindle axis</td>
</tr>
<tr>
<td>12</td>
<td>L M136 Feed rate in mm/rev</td>
</tr>
<tr>
<td>13</td>
<td>CYCL DEF 880 GEAR HOBBING Activate interpolation turning</td>
</tr>
</tbody>
</table>

Q215=+0 ;MACHINING OPERATION
Q540=+2.1 ;MODULE
Q541=+0 ;NUMBER OF TEETH
Q542=+69.3 ;OUTSIDE DIAMETER
Q543=+0.1666 ;TROUGH-TIP CLEARANCE
Q544=+5 ;ANGLE OF INCLINATION
Q545=+1.6833 ;TOOL LEAD ANGLE
Q546=+3 ;CHANGE TOOL DIRECTN.
Q550=+0 ;MACHINING SIDE
Q533=+0 ;PREFERRED DIRECTION
Q530=+2 ;INCLINED MACHINING
Q253=+2000 ;F PRE-POSITIONING
Q260=+20 ;CLEARANCE HEIGHT
Q553=+10 ;TOOL LENGTH OFFSET
Q551=+0 ;STARTING POINT IN Z
Q552=+10 ;END POINT IN Z
Q463=+1 ;MAX. CUTTING DEPTH
### 12.33 Example program

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>CYCL CALL M303</td>
<td>Call the cycle, spindle on</td>
</tr>
<tr>
<td>15</td>
<td>CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM</td>
<td>Reset the coordinate system</td>
</tr>
<tr>
<td>16</td>
<td>M145</td>
<td>Deactivate the active M144 in the cycle</td>
</tr>
<tr>
<td>17</td>
<td>FUNCTION MODE MILL</td>
<td>Activate milling mode</td>
</tr>
<tr>
<td>18</td>
<td>M140 MB MAX</td>
<td>Retract the tool in the tool axis</td>
</tr>
<tr>
<td>19</td>
<td>L A+0 C+0 R0 FMAX</td>
<td>Reset the rotation</td>
</tr>
<tr>
<td>20</td>
<td>M30</td>
<td>END of program</td>
</tr>
<tr>
<td>21</td>
<td>END PGM 5 MM</td>
<td>END of program</td>
</tr>
</tbody>
</table>
13

Using Touch Probe Cycles
13.1  General information about touch probe cycles

HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. Refer to your machine manual.

Method of function
Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further Information: Before You Start Working with Touch Probe Cycles, page 451
When the probe stylus contacts the workpiece,
- the 3-D touch probe transmits a signal to the TNC: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.
If the stylus is not deflected within a defined distance, the TNC displays an error message (distance: DIST from touch probe table).

Consideration of a basic rotation in the Manual Operation mode
During probing the TNC considers an active basic rotation and approaches the workpiece at an angle.

Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes
In the Manual Operation and El. Handwheel modes, the TNC provides touch probe cycles that allow you to:
- Calibrate the touch probe
- Compensating workpiece misalignment
- Setting datums
General information about touch probe cycles  13.1

**Touch probe cycles for automatic operation**

Besides the touch probe cycles, which you can use in the Manual and El. Handwheel modes, the TNC provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Datum setting
- Automatic workpiece inspection
- Automatic tool measurement

You can program the touch probe cycles in the Programming and Editing operating mode via the TOUCH PROBE key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the TNC shows a graphic during cycle definition. The graphic shows the parameter that needs to be entered (see figure at right).
Using Touch Probe Cycles

13.1 General information about touch probe cycles

Defining the touch probe cycle in the Programming and Editing mode of operation

- The soft-key row shows all available touch probe functions divided into groups.
- Select the desired probe cycle group, for example datum setting. Cycles for automatic tool measurement are available only if your machine has been prepared for them.
- Select a cycle, e.g. datum setting at pocket center. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the ENT key.
- The TNC ends the dialog when all required data has been entered.

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Group of measuring cycles</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cycles for automatic measurement and compensation of workpiece misalignment</td>
<td>458</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic workpiece presetting</td>
<td>480</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic workpiece inspection</td>
<td>534</td>
</tr>
<tr>
<td></td>
<td>Special cycles</td>
<td>580</td>
</tr>
<tr>
<td></td>
<td>Calibrate TS</td>
<td>580</td>
</tr>
<tr>
<td></td>
<td>Kinematics</td>
<td>619</td>
</tr>
<tr>
<td></td>
<td>Cycles for automatic tool measurement (enabled by the machine tool builder)</td>
<td>650</td>
</tr>
<tr>
<td></td>
<td>Visual Setup Control (Option 136 VSC)</td>
<td>596</td>
</tr>
</tbody>
</table>

NC blocks

```
5 TCH PROBE 410 DATUM INSIDE
RECTAN.
Q321=+50 ;CENTER IN 1ST AXIS
Q322=+50 ;CENTER IN 2ND AXIS
Q323=60 ;FIRST SIDE LENGTH
Q324=20 ;2ND SIDE LENGTH
Q261=5 ;MEASURING HEIGHT
Q320=0 ;SET-UP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=10 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+0 ;DATUM
```
13.2 Before You Start Working with Touch Probe Cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the path defined in DIST, the TNC outputs an error message.

Set-up clearance to touch point: SET_UP in touch probe table

In SET_UP you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to SET_UP.

Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use TRACK = ON to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

If you change TRACK = ON, you must recalibrate the touch probe.
Using Touch Probe Cycles

13.2 Before You Start Working with Touch Probe Cycles

Touch trigger probe, probing feed rate: $F$ in touch probe table
In $F$ you define the feed rate at which the TNC is to probe the workpiece.

Touch trigger probe, rapid traverse for positioning: $F_{\text{MAX}}$
In $F_{\text{MAX}}$ you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: $F_{\text{PREPOS}}$ in touch probe table
In $F_{\text{PREPOS}}$ you define whether the TNC is to position the touch probe at the feed rate defined in $F_{\text{MAX}}$ or at rapid traverse.
- Input value = $F_{\text{MAX}}_{\text{PROBE}}$: Position at feed rate from $F_{\text{MAX}}$
- Input value = $F_{\text{MAX}}_{\text{MACHINE}}$: Pre-position at rapid traverse
Multiple measurements

To increase measuring certainty, the TNC can run each probing process up to three times in sequence. Define the number of measurements in machine parameter ProbeSettings > Configuration of probe behavior > Automatic mode: Multiple measurements with probe function. If the measured position values differ too greatly, the TNC outputs an error message (the limit value is defined in Confidence interval of multiple measurements). With multiple measurement it is possible to detect random errors, e.g. from contamination.

If the measured values lie within the confidence interval, the TNC saves the mean value of the measured positions.

Confidence interval of multiple measurements

When you perform a multiple measurement, you store the value that the measured values may vary in ProbeSettings > Configuration of probe behavior > Automatic mode: Confidence interval of multiple measurements. If the difference in the measured values exceeds the value defined by you, the TNC outputs an error message.
Using Touch Probe Cycles

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.

![Danger of collision!]
When running touch probe cycles, no cycles must be active for coordinate transformation (Cycle 7 DATUM, Cycle 8 MIRROR IMAGE, Cycle 10 ROTATION, Cycles 11 SCALING and 26 AXIS-SPECIFIC SCALING).

You can also run the Touch Probe Cycles 408 to 419 during an active basic rotation. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT with datum tables after the measuring cycle.

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (defined in the cycle), the TNC retracts the touch probe in the probe axis to the clearance height and then positions it in the working plane to the first starting position.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the TNC first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height.
13.3 Touch probe table

General information
Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you run several touch probes on your machine tool, you can save separate data for each touch probe.

Editing touch probe tables
To edit the touch probe table, proceed as follows:
- Select the **Manual Operation** mode
- Select the touch probe functions: Press the **TOUCH PROBE** soft key. The TNC displays additional soft keys.
- Select the touch probe table: Press the **TCH PROBE TABLE** soft key.
- Set the **EDIT** soft key to **ON**
- Using the arrow keys, select the desired setting.
- Perform desired changes.
- Exit the touch probe table: Press the **END** soft key.
### Touch probe data

<table>
<thead>
<tr>
<th>Abbr.</th>
<th>Inputs</th>
<th>Dialog</th>
</tr>
</thead>
<tbody>
<tr>
<td>NO</td>
<td>Number of the touch probe: Enter this number in the tool table (column: TP_NO) under the appropriate tool number</td>
<td>–</td>
</tr>
<tr>
<td>TYPE</td>
<td>Selection of the touch probe used</td>
<td>Selection of touch probe?</td>
</tr>
<tr>
<td>CAL_OF1</td>
<td>Offset of the touch probe axis to the spindle axis for the reference axis</td>
<td>TS center misalign. ref. axis? [mm]</td>
</tr>
<tr>
<td>CAL_OF2</td>
<td>Offset of the touch probe axis to the spindle axis for the minor axis</td>
<td>TS center misalign. aux. axis? [mm]</td>
</tr>
<tr>
<td>CAL_ANG</td>
<td>The TNC orients the touch probe to the orientation angle before calibration or probing (if orientation is possible)</td>
<td>Spindle angle for calibration?</td>
</tr>
<tr>
<td>F</td>
<td>Feed rate at which the TNC is to probe the workpiece</td>
<td>Probing feed rate? [mm/min]</td>
</tr>
<tr>
<td>FMAX</td>
<td>Feed rate at which the touch probe pre-positions, or is positioned between the measuring points</td>
<td>Rapid traverse in probing cycle? [mm/min]</td>
</tr>
<tr>
<td>DIST</td>
<td>If the stylus is not deflected within the defined path, the TNC outputs an error message</td>
<td>Maximum measuring path? [mm]</td>
</tr>
<tr>
<td>SET_UP</td>
<td>In SET_UP you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to the SET_UP machine parameter.</td>
<td>Set-up clearance? [mm]</td>
</tr>
<tr>
<td>F_PREPOS</td>
<td>Defining speed with pre-positioning:</td>
<td>Pre-positioning at rap. traverse? ENT/NO ENT</td>
</tr>
<tr>
<td></td>
<td>- Pre-positioning with speed from FMAX: FMAX_PROBE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Pre-positioning with machine rapid traverse: FMAX_MACHINE</td>
<td></td>
</tr>
<tr>
<td>TRACK</td>
<td>To increase measuring accuracy, you can use TRACK = ON to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction:</td>
<td>Orient touch probe cycles? Yes=ENT, No=NOENT</td>
</tr>
<tr>
<td></td>
<td>- ON: Perform spindle tracking</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- OFF: Do not perform spindle tracking</td>
<td></td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment
### 14.1 Fundamentals

#### Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active. HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. Refer to your machine manual.

The TNC provides five cycles that enable you to measure and compensate workpiece misalignment. In addition, you can reset a basic rotation with Cycle 404:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>![400]</td>
<td>400 BASIC ROTATION</td>
<td>460</td>
</tr>
<tr>
<td></td>
<td>Automatic measurement using two points. Compensation via basic rotation.</td>
<td></td>
</tr>
<tr>
<td>![401]</td>
<td>401 ROT OF 2 HOLES</td>
<td>463</td>
</tr>
<tr>
<td></td>
<td>Automatic measurement using two holes. Compensation via basic rotation.</td>
<td></td>
</tr>
<tr>
<td>![402]</td>
<td>402 ROT OF 2 STUDS</td>
<td>466</td>
</tr>
<tr>
<td></td>
<td>Automatic measurement using two studs. Compensation via basic rotation.</td>
<td></td>
</tr>
<tr>
<td>![403]</td>
<td>403 ROT IN ROTARY AXIS</td>
<td>469</td>
</tr>
<tr>
<td></td>
<td>Automatic measurement using two points. Compensation by turning the table.</td>
<td></td>
</tr>
<tr>
<td>![405]</td>
<td>405 ROT IN C AXIS</td>
<td>473</td>
</tr>
<tr>
<td></td>
<td>Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.</td>
<td></td>
</tr>
<tr>
<td>![404]</td>
<td>404 SET BASIC ROTATION</td>
<td>472</td>
</tr>
<tr>
<td></td>
<td>Setting any basic rotation.</td>
<td></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 # (see figure at right). This enables you to measure the basic rotation against any straight line 1 of the workpiece and to establish the reference to the actual 0° direction 2.
14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

Cycle run

Touch probe cycle 400 determines a workpiece misalignment by measuring two points, which must lie on a straight surface. With the basic rotation function the TNC compensates the measured value.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) to the programmed touch point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F).

3. Then the touch probe moves to the next starting position 2 and probes the second position.

4. The TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC will reset an active basic rotation at the beginning of the cycle.
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative Traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>NC Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q263++10</td>
<td>1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264++3.5</td>
<td>1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q265++25</td>
<td>2ND POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q266++2</td>
<td>2ND POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q272=2</td>
<td>MEASURING AXIS</td>
</tr>
<tr>
<td>Q267=1</td>
<td>TRAVERSE DIRECTION</td>
</tr>
<tr>
<td>Q261=5</td>
<td>MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0</td>
<td>SET-UP 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>Q307=0</td>
<td>PRESET ROT. ANGLE</td>
</tr>
<tr>
<td>Q305=0</td>
<td>NO. IN TABLE</td>
</tr>
</tbody>
</table>
14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points

- **Preset value for rotation angle** Q307 (absolute):
  If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000

- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. Input range 0 to 99999
14.3 BASIC ROTATION over two holes  
(Cycle 401, DIN/ISO: G401)

Cycle run

The Touch Probe Cycle 401 measures the centers of two holes. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the hole centers. With the basic rotation function, the TNC compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to the center of the first hole 1.

2 Then the probe moves to the entered measuring height and probes four points to find the first hole center.

3 The touch probe returns to the clearance height and then to the position entered as center of the second hole 2.

4 The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.

5 Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC will reset an active basic rotation at the beginning of the cycle. If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X
14.3 BASIC ROTATION over two holes (Cycle 401, DIN/ISO: G401)

**Cycle parameters**

- **1st hole: Center in 1st axis** Q268 (absolute): Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st hole: Center in 2nd axis** Q269 (absolute): Center of the first hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 1st axis** Q270 (absolute): Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 2nd axis** Q271 (absolute): Center of the second hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Preset value for rotation angle** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (Q402=1). In this case the misalignment is not saved as an angular value. Input range 0 to 99999

**NC blocks**

```
5 TCH PROBE 401 ROT OF 2 HOLES
Q268=-37 ;1ST CENTER IN 1ST AXIS
Q269=+12 ;1ST CENTER 2ND AXIS
Q270=+75 ;2ND CENTER 1ST AXIS
Q271=+20 ;2ND CENTER 2ND AXIS
Q261=-5 ;MEASURING HEIGHT
Q260=+20 ;CLEARANCE HEIGHT
Q307=0 ;PRESET ROT. ANGLE
Q305=0 ;NO. IN TABLE
Q402=0 ;COMPENSATION
Q337=0 ;ZERO RESET
```
Compensation Q402: Define whether the TNC should set the measured misalignment as basic rotation or should align via rotating the rotary table:

- 0: Set basic rotation
- 1: Rotate the rotary table

If you specify rotating the rotary table, the TNC does not save the measured misalignment, even if you have defined a table row in parameter Q305.

Set to zero after alignment Q337: Define whether the TNC should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment:

- 0: Do not set the angle of the rotary axis to 0 in the table after alignment
- 1: Set the angle of the rotary axis to 0 in the table after alignment. The TNC sets the display to 0 only if you have defined Q402=1.
14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402)

**Cycle run**

The Touch Probe Cycle 402 measures the centers of two studs. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the two stud centers. With the basic rotation function, the TNC compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1 of the first stud.
2. Then the probe moves to the entered measuring height 1 and probes four points to find the center of the first stud. The touch probe moves on a circular arc between the touch points, each of which is offset by 90°.
3. The touch probe returns to the clearance height and then positions the probe to starting point 5 of the second stud.
4. The probe moves to the entered measuring height 2 and probes four points to find the center of the second stud.
5. Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

**Please note while programming:**

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC will reset an active basic rotation at the beginning of the cycle. If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X
Cycle parameters

- **1st stud: Center in 1st axis** Q268 (absolute): Center of the first stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st stud: Center in 2nd axis** Q269 (absolute): Center of the first stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Diameter of stud 1** Q313: Approximate diameter of the 1st stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Measuring height 1 in the probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 1 is to be measured. Input range -99999.9999 to 99999.9999
- **2nd stud: Center in 1st axis** Q270 (absolute): Center of the second stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd stud: Center in 2nd axis** Q271 (absolute): Center of the second stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Diameter of stud 2** Q314: Approximate diameter of the 2nd stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Measuring height of stud 2 in the probe axis** Q315 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 2 is to be measured. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 402 ROT OF 2 STUDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q268=-37 ; 1ST CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q269=+12 ; 1ST CENTER 2ND AXIS</td>
</tr>
<tr>
<td>Q313=60 ; DIAMETER OF STUD 1</td>
</tr>
<tr>
<td>Q261=-5 ; MEASURING HEIGHT 1</td>
</tr>
<tr>
<td>Q270=+75 ; 2ND CENTER 1ST AXIS</td>
</tr>
<tr>
<td>Q271=+20 ; 2ND CENTER 2ND AXIS</td>
</tr>
<tr>
<td>Q314=60 ; DIAMETER OF STUD 2</td>
</tr>
<tr>
<td>Q315=+5 ; MEASURING HEIGHT 2</td>
</tr>
<tr>
<td>Q320=0 ; SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260++20 ; CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ; MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q307=0 ; PRESET ROT. ANGLE</td>
</tr>
<tr>
<td>Q305=0 ; NO. IN TABLE</td>
</tr>
<tr>
<td>Q402=0 ; COMPENSATION</td>
</tr>
<tr>
<td>Q337=0 ; ZERO RESET</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.4 BASIC ROTATION over two studs (Cycle 402, DIN/ISO: G402)

- **Preset value for rotation angle** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000

- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (Q402=1). In this case the misalignment is not saved as an angular value. Input range 0 to 99999

- **Compensation** Q402: Define whether the TNC should set the measured misalignment as basic rotation or should align via rotating the rotary table:
  - 0: Set basic rotation
  - 1: Rotate the rotary table
  If you specify rotating the rotary table, the TNC does not save the measured misalignment, even if you have defined a table row in parameter Q305.

- **Set to zero after alignment** Q337: Define whether the TNC should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment:
  - 0: Do not set the angle of the rotary axis to 0 in the table after alignment
  - 1: Set the angle of the rotary axis to 0 in the table after alignment. The TNC sets the display to 0 only if you have defined Q402=1.
14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403)

Cycle run

Touch probe cycle 403 determines a workpiece misalignment by measuring two points, which must lie on a straight line. The TNC compensates the determined misalignment by rotating the A, B or C axis. The workpiece can be clamped in any position on the rotary table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \( \text{FMAX} \) column) (see "Executing touch probe cycles", page 454) to the programmed touch point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \( \text{F} \)).

3. Then the touch probe moves to the next starting position 2 and probes the second position.

4. The TNC returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally you can specify whether the TNC is to set the determined rotation angle to 0 in the preset table or in the datum table.

Please note while programming:

**Danger of collision!**

Ensure that the clearance height is sufficiently large so that no collisions can occur during the final positioning of the rotary axis.

If you enter 0 in parameter Q312 Axis for compensating movement, the cycle automatically determines the rotary axis to be aligned (recommended setting). Depending on the sequence of the probing points, an angle with the actual direction is determined. The measured angle goes from the first to the second probing point. If you select the A, B or C axis as compensation axis in parameter Q312, the cycle determines the angle, regardless of the sequence of the probing points. The calculated angle lies in the range from \(-90^\circ\) to \(+90^\circ\). After alignment, check the position of the rotary axis.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC stores the measured angle in parameter Q150.
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403)

Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis (1…3: 1 = principal axis)** Q272: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction** 1 Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative Traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999

NC blocks

```
5 TCH PROBE 403 ROT IN ROTARY AXIS
Q263=+0 ;1ST POINT 1ST AXIS
Q264=+0 ;1ST POINT 2ND AXIS
Q265=+20 ;2ND POINT 1ST AXIS
Q266=+30 ;2ND POINT 2ND AXIS
Q272=1 ;MEASURING AXIS
Q267=-1 ;TRAVERSE DIRECTION
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SET-UP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
```
BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403)

- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

- **Axis for compensation movement** Q312:
  Assignment of the rotary axis in which the TNC is to compensate the measured misalignment:
  0: Automatic mode – the TNC uses the active kinematics to determine the rotary axis to be aligned. In automatic mode the first rotary table axis (as viewed from the workpiece) is used as compensation axis. Recommended setting.
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C

- **Set to zero after alignment** Q337: Define whether the TNC should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment:
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment

- **Number in table** Q305: Enter the number in the preset table/datum table in which the TNC is to set the rotary axis to zero. Only effective if Q337 is set to 1. Input range 0 to 99999

- **Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as datum shift active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).

- **Reference angle? (0=ref. axis)** Q380: Angle with which the TNC is to align the probed straight line. Only effective if rotary axis = automatic mode is selected, or rotary axis = C is selected (Q312 = 0 or 6). Input range -360.000 to 360.000

| Q312=0 | COMPENSATION AXIS |
| Q337=0 | ZERO RESET |
| Q305=1 | NO. IN TABLE |
| Q303++1 | MEAS. VALUE TRANSFER |
| Q380++90 | REFERENCE ANGLE |
14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)

Cycle run
With Touch Probe Cycle 404, during program run you can automatically set any basic rotation or save it to the preset table. You can also use Cycle 404 if you want to reset an active basic rotation.

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 404 BASIC ROTATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q307=+0 ;PRESET ROTATION ANG.</td>
</tr>
<tr>
<td>Q305=-1 ;NO. IN TABLE</td>
</tr>
</tbody>
</table>

Cycle parameters
- **Preset value for rotation angle**: Angular value at which the basic rotation is to be set. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. Input range -1 to 99999. If you enter Q305=0 or Q305=1, the TNC additionally saves the determined basic rotation in the basic rotation menu (PROBING ROT) of the Manual Operation mode.
- **-1** = Overwrite and activate the active preset
- **0** = Copy the active preset to preset line 0, write the basic rotation to preset line 0 and activate preset 0
- **>1** = Save the basic rotation to the specified preset. The preset is not activated.
Compensating workpiece misalignment by rotating the C axis

14.7 Compensating workpiece misalignment by rotating the C axis
(Cycle 405, DIN/ISO: G405)

Cycle run

With Touch Probe Cycle 405, you can measure
- the angular offset between the positive Y axis of the active coordinate system and the center of a hole, or
- the angular offset between the nominal position and the actual position of a hole center.

The TNC compensates the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive.

If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.

3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the touch probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points and positions the touch probe on the hole center measured.

5. Finally the TNC returns the touch probe to the clearance height and aligns the workpiece by rotating the table. The TNC rotates the rotary table so that the hole center after compensation lies in the direction of the positive Y axis, or on the nominal position of the hole center—both with a vertical and horizontal touch probe axis. The measured angular misalignment is also available in parameter Q150.
14.7 Compensating workpiece misalignment by rotating the C axis (Cycle 405, DIN/ISO: G405)

Please note while programming:

**Danger of collision!**

To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the circle center. Minimum input value: 5°.
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
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 405 ROT IN C AXIS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q321=+50 ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q322=+50 ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q262=10 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q325=+0 ;STARTING ANGLE</td>
</tr>
<tr>
<td>Q247=90 ;STEPPING ANGLE</td>
</tr>
<tr>
<td>Q261=+5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q337=0 ;ZERO RESET</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.7 Compensating workpiece misalignment by rotating the C axis
(Cycle 405, DIN/ISO: G405)

- **Set to zero after alignment** Q337: definition of whether the TNC should set the display of the C-axis to zero, or write the angular misalignment in column C of the datum table:
  - 0: Set the display of the C-axis to zero
  - >0: Write the measured angular misalignment with correct algebraic signs in the datum table.
  Line number = value of Q337. If a C-axis shift is registered in the datum table, the TNC adds the measured angular misalignment.
Example: Determining a basic rotation from two holes

### 14.8 Example: Determining a basic rotation from two holes

<table>
<thead>
<tr>
<th>Step</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 OF 2 HOLES</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>Q268=+25 ; 1ST CENTER 1ST AXIS</td>
<td>Center of the 1st hole: X coordinate</td>
</tr>
<tr>
<td>15</td>
<td>Q269=+15 ; 1ST CENTER 2ND AXIS</td>
<td>Center of the 1st hole: Y coordinate</td>
</tr>
<tr>
<td>80</td>
<td>Q270=+80 ; 2ND CENTER 1ST AXIS</td>
<td>Center of the 2nd hole: X coordinate</td>
</tr>
<tr>
<td>35</td>
<td>Q271=+35 ; 2ND CENTER 2ND AXIS</td>
<td>Center of the 2nd hole: Y coordinate</td>
</tr>
<tr>
<td>5</td>
<td>Q261=-5 ; MEASURING HEIGHT</td>
<td>Coordinate in the touch probe axis in which the measurement is made</td>
</tr>
<tr>
<td>20</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>0</td>
<td>Q307=+0 ; PRESET ROT. ANGLE</td>
<td>Angle of the reference line</td>
</tr>
<tr>
<td>1</td>
<td>Q402=1 ; COMPENSATION</td>
<td>Compensate misalignment by rotating the rotary table</td>
</tr>
<tr>
<td>1</td>
<td>Q337=1 ; ZERO RESET</td>
<td>Set the display to zero after the alignment</td>
</tr>
<tr>
<td>3</td>
<td>CALL PGM 35K47</td>
<td>Call part program</td>
</tr>
<tr>
<td>4</td>
<td>END PGM CYC401 MM</td>
<td></td>
</tr>
</tbody>
</table>
15

Touch Probe Cycles: Automatic Datum Setting
15.1 Fundamentals

Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active. HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. Refer to your machine manual.

The TNC offers twelve cycles for automatically finding reference points and handling them as follows:

- Setting the determined values directly as display values
- Entering the determined values in the preset table
- Entering the determined values in a datum table
<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>408</td>
<td>408 SLOT CENTER REF PT.</td>
<td>484</td>
</tr>
<tr>
<td></td>
<td>Measuring the inside width of a slot, and defining the slot center as datum</td>
<td></td>
</tr>
<tr>
<td>409</td>
<td>409 RIDGE CENTER REF PT.</td>
<td>488</td>
</tr>
<tr>
<td></td>
<td>Measuring the outside width of a ridge, and defining the ridge center as datum</td>
<td></td>
</tr>
<tr>
<td>410</td>
<td>410 DATUM INSIDE RECTANGLE</td>
<td>491</td>
</tr>
<tr>
<td></td>
<td>Measuring the inside length and width of a rectangle, and defining the center as datum</td>
<td></td>
</tr>
<tr>
<td>411</td>
<td>411 DATUM OUTSIDE RECTANGLE</td>
<td>495</td>
</tr>
<tr>
<td></td>
<td>Measuring the outside length and width of a rectangle, and defining the center as datum</td>
<td></td>
</tr>
<tr>
<td>412</td>
<td>412 DATUM INSIDE CIRCLE</td>
<td>498</td>
</tr>
<tr>
<td></td>
<td>Measuring any four points on the inside of a circle, and defining the center as datum</td>
<td></td>
</tr>
<tr>
<td>413</td>
<td>413 DATUM OUTSIDE CIRCLE</td>
<td>503</td>
</tr>
<tr>
<td></td>
<td>Measuring any four points on the outside of a circle, and defining the center as datum</td>
<td></td>
</tr>
<tr>
<td>414</td>
<td>414 DATUM OUTSIDE CORNER</td>
<td>507</td>
</tr>
<tr>
<td></td>
<td>Measuring two lines from the outside of the angle, and defining the intersection as datum</td>
<td></td>
</tr>
<tr>
<td>415</td>
<td>415 DATUM INSIDE CORNER</td>
<td>512</td>
</tr>
<tr>
<td></td>
<td>Measuring two lines from within the angle, and defining the intersection as datum</td>
<td></td>
</tr>
<tr>
<td>416</td>
<td>416 DATUM CIRCLE CENTER</td>
<td>516</td>
</tr>
<tr>
<td></td>
<td>(2nd soft-key level) Measuring any three holes on a bolt hole circle, and defining the bolt-hole center as datum</td>
<td></td>
</tr>
<tr>
<td>417</td>
<td>417 DATUM IN TS AXIS</td>
<td>520</td>
</tr>
<tr>
<td></td>
<td>(2nd soft-key level) Measuring any position in the touch probe axis and defining it as datum</td>
<td></td>
</tr>
<tr>
<td>418</td>
<td>418 DATUM FROM 4 HOLES</td>
<td>522</td>
</tr>
<tr>
<td></td>
<td>(2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum</td>
<td></td>
</tr>
<tr>
<td>419</td>
<td>419 DATUM IN ONE AXIS</td>
<td>526</td>
</tr>
<tr>
<td></td>
<td>(2nd soft-key row) Measuring any position in any axis and defining it as datum</td>
<td></td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Datum Setting

15.1 Fundamentals

Characteristics common to all touch probe cycles for datum setting

You can also run the Touch Probe Cycles 408 to 419 during an active rotation (basic rotation or Cycle 10).

Datum point and touch probe axis
From the touch probe axis that you have defined in the measuring program the TNC determines the working plane for the datum.

<table>
<thead>
<tr>
<th>Active touch probe axis</th>
<th>Datum setting in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>X and Y</td>
</tr>
<tr>
<td>Y</td>
<td>Z and X</td>
</tr>
<tr>
<td>X</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 reference point in the active datum table. The reference system is the active workpiece coordinate system. The value of parameter Q305 determines the datum number. **Activate the datum with Cycle 7 in the part program.**

- **Q305 not equal to 0, Q303 = 1** The TNC writes the calculated reference point in the preset table. The reference system is the machine coordinate system (REF coordinates). The value of parameter Q305 determines the preset number. **Activate the preset with Cycle 247 in the part program.**

**Measurement results in Q parameters**
The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. You can use these parameters in your program. Note the table of result parameters listed with every cycle description.
15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408)

Cycle run

Touch Probe Cycle 408 finds the center of a slot and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \( F_{\text{MAX}} \) column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the \( \text{SET}_\text{UP} \) column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \( F \)).

3. Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see “Characteristics common to all touch probe cycles for datum setting”, page 482) 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.
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
15.2 DATUM SLOT CENTER (Cycle 408, DIN/ISO: G408)

Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the slot in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the slot in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Width of slot** Q311 (incremental): Width of the slot, regardless of its position in the working plane. Input range 0 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the slot center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the slot center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated slot center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as datum shift into 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).

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 408 SLOT CENTER REF PT</td>
</tr>
<tr>
<td>Q321++50  ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q322++50  ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q311+25   ;SLOT WIDTH</td>
</tr>
<tr>
<td>Q272=1    ;MEASURING AXIS</td>
</tr>
<tr>
<td>Q261=-5   ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0    ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260++20  ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0    ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q305=10   ;NO. IN TABLE</td>
</tr>
<tr>
<td>Q405++0   ;DATUM</td>
</tr>
<tr>
<td>Q303++1   ;MEAS. VALUE TRANSFER</td>
</tr>
<tr>
<td>Q381=1    ;PROBE IN TS AXIS</td>
</tr>
<tr>
<td>Q382++85  ;1ST CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q383++50  ;2ND CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q384++0   ;3RD CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q333++1   ;DATUM</td>
</tr>
</tbody>
</table>
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis

- **Probe TS axis: Coord. 1st axis** Q382 (absolute):
  Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

- **Probe TS axis: Coord. 2nd axis** Q383 (absolute):
  Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **Probe TS axis: Coord. 3rd axis** Q384 (absolute):
  Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
Touch Probe Cycles: Automatic Datum Setting

15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409)

Cycle run

Touch Probe Cycle 409 finds the center of a ridge and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from $F_{\text{MAX}}$ column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the $SET\_UP$ column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column $F$).

3. Then the touch probe moves at clearance height to the next touch point 2 and probes the second touch point.

4. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see “Characteristics common to all touch probe cycles for datum setting”, page 482) 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. If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the ridge in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the ridge in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Width of ridge** Q311 (incremental): Width of the ridge, regardless of its position in the working plane. Input range 0 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the ridge center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the ridge center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated ridge center. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as datum shift active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis

NC blocks

| Q321=+50 | CENTER IN 1ST AXIS |
| Q322=+50 | CENTER IN 2ND AXIS |
| Q311=25 | SLOT WIDTH |
| Q272=1 | MEASURING AXIS |
| Q261=5 | MEASURING HEIGHT |
| Q320=0 | SET-UP CLEARANCE |
| Q260=+20 | CLEARANCE HEIGHT |
| Q305=10 | NO. IN TABLE |
| Q405=+0 | DATUM |
| Q303=+1 | MEAS. VALUE TRANSFER |
| Q381=1 | PROBE IN TS AXIS |
| Q382=+85 | 1ST CO. FOR TS AXIS |
| Q383=+50 | 2ND CO. FOR TS AXIS |
| Q384=+0 | 3RD CO. FOR TS AXIS |
| Q333=+1 | DATUM |
15.3 DATUM RIDGE CENTER (Cycle 409, DIN/ISO: G409)

- **Probe TS axis: Coord. 1st axis Q382 (absolute):**
  Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

- **Probe TS axis: Coord. 2nd axis Q383 (absolute):**
  Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **Probe TS axis: Coord. 3rd axis Q384 (absolute):**
  Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **New datum in TS axis Q333 (absolute):**
  Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

Cycle run

Touch Probe Cycle 410 finds the center of a rectangular pocket and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \textit{FMAX} column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the \textit{SET\_UP} column of the touch probe table.

2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \textit{F}).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4 The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482).

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>
Touch Probe Cycles: Automatic Datum Setting

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

Please note while programming:

**Danger of collision!**
To prevent a collision between touch probe and workpiece, enter low estimates for the lengths of the first and second sides.

If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
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
- **1st side length** Q323 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q321=+50 ; CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q322=+50 ; CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q323=60 ; FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q324=20 ; 2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q261=5 ; MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ; SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20 ; CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ; MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q305=10 ; NO. IN TABLE</td>
</tr>
<tr>
<td>Q311=+0 ; DATUM</td>
</tr>
<tr>
<td>Q332=+0 ; DATUM</td>
</tr>
<tr>
<td>Q303=+1 ; MEAS. VALUE TRANSFER</td>
</tr>
<tr>
<td>Q381=1 ; PROBE IN TS AXIS</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Datum Setting

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

- **New datum for reference axis** Q331 (absolute):
  Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0.
  Input range -99999.9999 to 99999.9999

- **New datum for minor axis** Q332 (absolute):
  Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0
  Input range -99999.9999 to 99999.9999

- **Measured-value transfer (0, 1)** Q303:
  Specify whether the determined datum is to be saved in the datum table or in the preset table:
  - 1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 482)
  - 0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
  - 1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381:
  Specify whether the TNC should also set the datum in the touch probe axis:
  - 0: Do not set the datum in the touch probe axis
  - 1: Set the datum in the touch probe axis

- **Probe TS axis: Coord. 1st axis** Q382 (absolute):
  Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 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** Q333 (absolute):
  Coordinate at which the TNC should set the datum. Default setting = 0
  Input range -99999.9999 to 99999.9999

<table>
<thead>
<tr>
<th>Q382=+85</th>
<th>1ST CO. FOR TS AXIS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q383=+50</td>
<td>2ND CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q384=+0</td>
<td>3RD CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q333=+1</td>
<td>DATUM</td>
</tr>
</tbody>
</table>
15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411)

Cycle run

Touch Probe Cycle 411 finds the center of a rectangular stud and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see “Executing touch probe cycles”, page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F).

3. Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see “Characteristics common to all touch probe cycles for datum setting”, page 482).

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 high estimates for the lengths of the 1st and 2nd sides.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
Touch Probe Cycles: Automatic Datum Setting

15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411)

**Cycle parameters**

- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q323 (incremental): Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the stud center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **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

**NC blocks**

<table>
<thead>
<tr>
<th>5 TCH PROBE 411 DATUM OUTS. RECTAN.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q321=+50 ;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q322=+50 ;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q323=60 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q324=20 ;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q305=0 ;NO. IN TABLE</td>
</tr>
<tr>
<td>Q331=+0 ;DATUM</td>
</tr>
<tr>
<td>Q332=+0 ;DATUM</td>
</tr>
<tr>
<td>Q381=1 ;PROBE IN TS AXIS</td>
</tr>
<tr>
<td>Q303=+1 ;MEAS. VALUE TRANSFER</td>
</tr>
</tbody>
</table>
Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 482)
0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

Probe in TS axis Q381: Specify whether the TNC should also set the datum in the touch probe axis:
0: Do not set the datum in the touch probe axis
1: Set the datum in the touch probe axis

Probe TS axis: Coord. 1st axis Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 2nd axis Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 3rd axis Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

New datum in TS axis Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
Touch Probe Cycles: Automatic Datum Setting

15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

Cycle run

Touch Probe Cycle 412 finds the center of a circular pocket (or of a hole) and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from $F_{MAX}$ column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column $F$). The TNC derives the probing direction automatically from the programmed starting angle.

3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482) 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.
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the pocket in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999

NC blocks

```
5 TCH PROBE 412 DATUM INSIDE CIRCLE
Q321=+50 ;CENTER IN 1ST AXIS
Q322=+50 ;CENTER IN 2ND AXIS
Q262=75 ;NOMINAL DIAMETER
Q325=+0 ;STARTING ANGLE
Q247=+60 ;STEPPING ANGLE
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SET-UP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=12 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;DATUM
Q423=4 ;NO. OF PROBE POINTS
Q365=1 ;TYPE OF TRAVERSE
```
New datum for reference axis Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999

New datum for minor axis Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use! Is entered by the TNC when old programs are read in (see “Characteristics common to all touch probe cycles for datum setting”, page 482)
0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

Probe in TS axis Q381: Specify whether the TNC should also set the datum in the touch probe axis:
0: Do not set the datum in the touch probe axis
1: Set the datum in the touch probe axis

Probe TS axis: Coord. 1st axis Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 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
15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  - 4: Use 4 measuring points (default setting)
  - 3: Use 3 measuring points

- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if “traverse to clearance height” (Q301=1) is active:
  - 0: Move in a straight line between machining operations
  - 1: Move in a circular arc on the pitch circle diameter between machining operations
15.7 DATUM FROM OUTSIDE OF CIRCLE  
(Cycle 413, DIN/ISO: G413)

**Cycle run**

Touch Probe Cycle 413 finds the center of a circular stud and defines it as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \(F_{MAX}\) column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \(F\)). The TNC derives the probing direction automatically from the programmed starting angle.

3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482) 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°.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
Touch Probe Cycles: Automatic Datum Setting

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)

Cycle parameters

- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
<th>5 TCH PROBE 413 DATUM OUTSIDE CIRCLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q321=+50</td>
<td>;CENTER IN 1ST AXIS</td>
</tr>
<tr>
<td>Q322=+50</td>
<td>;CENTER IN 2ND AXIS</td>
</tr>
<tr>
<td>Q262=75</td>
<td>;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q325=+0</td>
<td>;STARTING ANGLE</td>
</tr>
<tr>
<td>Q247=+60</td>
<td>;STEPPING ANGLE</td>
</tr>
<tr>
<td>Q261=+5</td>
<td>;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0</td>
<td>;SET-UP 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>Q305=15</td>
<td>;NO. IN TABLE</td>
</tr>
<tr>
<td>Q331=+0</td>
<td>;DATUM</td>
</tr>
<tr>
<td>Q332=+0</td>
<td>;DATUM</td>
</tr>
</tbody>
</table>
**Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the stud center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999

**New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0 input range -99999.9999 to 99999.9999

**New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0 input range -99999.9999 to 99999.9999

**Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
- 1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 482)
- 0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
- 1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

**Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
- 0: Do not set the datum in the touch probe axis
- 1: Set the datum in the touch probe axis

**Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st. Input range -99999.9999 to 99999.9999

**Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

<table>
<thead>
<tr>
<th>Q303</th>
<th>MEAS. VALUE TRANSFER</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q381</td>
<td>PROBE IN TS AXIS</td>
</tr>
<tr>
<td>Q382</td>
<td>1ST CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q383</td>
<td>2ND CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q384</td>
<td>3RD CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q333</td>
<td>DATUM</td>
</tr>
<tr>
<td>Q423</td>
<td>NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q365</td>
<td>TYPE OF TRAVERSE</td>
</tr>
</tbody>
</table>
Probe TS axis: Coord. 3rd axis Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

New datum in TS axis Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

No. of measuring points (4/3) Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
4: Use 4 measuring points (default setting)
3: Use 3 measuring points

Type of traverse? Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
0: Move in a straight line between machining operations
1: Move in a circular arc on the pitch circle diameter between machining operations
15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414)

**Cycle run**

Touch Probe Cycle 414 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1 (see figure at upper right). The TNC offsets the touch probe by the safety clearance in the direction opposite to the respective traverse direction.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column $F$). The TNC derives the probing direction automatically from the programmed 3rd measuring point.

3. Then the touch probe moves to the next starting position and from there probes the second position.

4. The TNC positions the probe to starting point and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482) and saves the coordinates of the determined corner in the Q parameters listed below.

4. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

<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:

**Danger of collision!**

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC always measures the first line in the direction of the minor axis of the working plane. By defining the positions of the measuring points 1 and 3 you also determine the corner at which the TNC sets the datum (see figure at right and table below).

<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
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points

---

**NC blocks**

<table>
<thead>
<tr>
<th>5 TCH PROBE 414 DATUM INSIDE CORNER</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q263=+37 ;1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264=+7 ;1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q265=+95 ;2ND POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q266=+25 ;2ND POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q267=+45 ;SPACING IN 1ST AXIS</td>
</tr>
<tr>
<td>Q268=+50 ;SPACING IN 2ND AXIS</td>
</tr>
<tr>
<td>Q269=+5 ;MEASURING HEIGTH</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q302=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q303=0 ;NO. IN TABLE</td>
</tr>
<tr>
<td>Q304=0 ;BASIC ROTATION</td>
</tr>
<tr>
<td>Q305=0 ;DATUM</td>
</tr>
</tbody>
</table>
15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414)

- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation

- **Datum number in table** Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999

- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0 input range -99999.9999 to 99999.9999

- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0 input range -99999.9999 to 99999.9999

- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 482)
  0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set the datum in the touch probe axis
  1: Set the datum in the touch probe axis

- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

---

<table>
<thead>
<tr>
<th>Q332=+0 ;DATUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q303=+1 ;MEAS. VALUE TRANSFER</td>
</tr>
<tr>
<td>Q381=1 ;PROBE IN TS AXIS</td>
</tr>
<tr>
<td>Q382=+85 ;1ST CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q383=+50 ;2ND CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q384=+0 ;3RD CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q333=+1 ;DATUM</td>
</tr>
</tbody>
</table>
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
**15.9 DATUM FROM INSIDE OF CORNER** (Cycle 415, DIN/ISO: G415)

**Cycle run**

Touch Probe Cycle 415 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from $F_{MAX}$ column) (see "Executing touch probe cycles", page 454) to touch point 1 (see figure at upper right) that you have defined in the cycle. The TNC offsets the touch probe by the safety clearance in the direction opposite to the respective traverse direction.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column $F$). The probing direction is derived from the number by which you identify the corner.

1. Then the touch probe moves to the next starting position 2 and from there probes the second position.

2. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

3. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482) and saves the coordinates of the determined corner in the Q parameters listed below.

4. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

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

Please note while programming:

Danger of collision!
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
The TNC always measures the first line in the direction of the minor axis of the working plane.
15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415)

Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- **Corner** Q308: Number identifying the corner which the TNC is to set as datum. Input range 1 to 4
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  - 0: Do not execute basic rotation
  - 1: Execute basic rotation
- **Datum number in table** Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 415 DATUM OUTSIDE CORNER</td>
</tr>
<tr>
<td>Q263=-37 ;1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264=-7 ;1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q326=50 ;SPACING IN 1ST AXIS</td>
</tr>
<tr>
<td>Q296=-95 ;3RD POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q297=-25 ;3RD POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q327=45 ;SPACING IN 2ND AXIS</td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=-20 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q304=0 ;BASIC ROTATION</td>
</tr>
<tr>
<td>Q305=7 ;NO. IN TABLE</td>
</tr>
<tr>
<td>Q311=-0 ;DATUM</td>
</tr>
<tr>
<td>Q332=-0 ;DATUM</td>
</tr>
<tr>
<td>Q303=-1 ;MEAS. VALUE TRANSFER</td>
</tr>
<tr>
<td>Q381=1 ;PROBE IN TS AXIS</td>
</tr>
<tr>
<td>Q382=-85 ;1ST CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q383=-50 ;2ND CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q384=0 ;3RD CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q333=-1 ;DATUM</td>
</tr>
</tbody>
</table>
New datum for reference axis Q331 (absolute):
Coordinate in the reference axis at which the TNC
should set the corner. Default setting = 0 input
range -99999.9999 to 99999.9999

New datum for minor axis Q332 (absolute):
Coordinate in the minor axis at which the TNC
should set the calculated corner. Default setting = 0
input range -99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify
whether the determined datum is to be saved in the
datum table or in the preset table:
-1: Do not use! Is entered by the TNC when old
programs are read in (see "Characteristics common
to all touch probe cycles for datum setting",
page 482)
0: Write the measured datum into the active datum
table. The reference system is the active workpiece
coordinate system
1: Write the measured datum into the preset table.
The reference system is the machine coordinate
system (REF system).

Probe in TS axis Q381: Specify whether the TNC
should also set the datum in the touch probe axis:
0: Do not set the datum in the touch probe axis
1: Set the datum in the touch probe axis

Probe TS axis: Coord. 1st axis Q382 (absolute):
Coordinate of the probe point in the reference axis
of the working plane at which point the datum is
to be set in the touch probe axis. Only effective if
Q381 = 1st input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 2nd axis Q383 (absolute):
Coordinate of the probe point in the minor axis of
the working plane at which point the datum is to
be set in the touch probe axis. Only effective if Q381 =
1. Input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 3rd axis Q384 (absolute):
Coordinate of the probe point in the touch probe
axis, at which point the datum is to be set in the
touch probe axis. Only effective if Q381 = 1. Input
range -99999.9999 to 99999.9999

New datum in TS axis Q333 (absolute): Coordinate
in the touch probe axis at which the TNC should
set the datum. Default setting = 0. Input range
-99999.9999 to 99999.9999
Touch Probe Cycles: Automatic Datum Setting

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

**Cycle run**

Touch Probe Cycle 416 finds the center of a bolt hole circle and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \textit{FMAX} column) (see "Executing touch probe cycles", page 454) to the center of the first hole 1.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole 2.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The touch probe returns to the clearance height and then to the position entered as center of the third hole 3.
6. The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482) 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:

**Danger of collision!**
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
**Touch Probe Cycles: Automatic Datum Setting**

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

**Cycle parameters**

- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input range 0 to 99999.9999
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the bolt-hole circle center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the bolt-hole circle center. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the bolt-hole center. Default setting = 0 input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the bolt-hole center. Default setting = 0 input range -99999.9999 to 99999.9999

**NC blocks**

```
5 TCH PROBE 416 DATUM CIRCLE CENTER
Q273=+50 ;CENTER IN 1ST AXIS
Q274=+50 ;CENTER IN 2ND AXIS
Q262=90 ;NOMINAL DIAMETER
Q291=+34 ;ANGLE OF 1ST HOLE
Q292=+70 ;ANGLE OF 2ND HOLE
Q293=+210 ;ANGLE OF 3RD HOLE
Q261=-5 ;MEASURING HEIGHT
Q260=+20 ;CLEARANCE HEIGHT
Q305=12 ;NO. IN TABLE
Q331=+0 ;DATUM
Q332=+0 ;DATUM
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
```
Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use! Is entered by the TNC when old programs are read in (see “Characteristics common to all touch probe cycles for datum setting”, page 482)
0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

Probe in TS axis Q381: Specify whether the TNC should also set the datum in the touch probe axis:
0: Do not set the datum in the touch probe axis
1: Set the datum in the touch probe axis

Probe TS axis: Coord. 1st axis Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 2nd axis Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 3rd axis Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

New datum in TS axis Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

Set-up clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table), and is only effective when the datum is probed in the touch probe axis. Input range 0 to 99999.9999
15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417)

**Cycle run**

Touch Probe Cycle 417 measures any coordinate in the touch probe axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from $F_{MAX}$ column) (see "Executing touch probe cycles", page 454) to the programmed touch point 1. The TNC offsets the touch probe by the safety clearance in the positive direction of the touch probe axis.

2. Then the touch probe moves in its own axis to the coordinate entered as starting point 1 and measures the actual position with a simple probing movement.

3. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482) and saves the actual value 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:

- **Danger of collision!**
  If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

- Before a cycle definition you must have programmed a tool call to define the touch probe axis.
  The TNC then sets the datum in this axis.
DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417) 15.11

Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 3rd axis** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0 input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  - 0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
  - 1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 417 DATUM IN TS AXIS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q263=+25 ;1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264=+25 ;1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q294=+25 ;1ST POINT 3RD AXIS</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+50 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q305=0 ;NO. IN TABLE</td>
</tr>
<tr>
<td>Q333=+0 ;DATUM</td>
</tr>
<tr>
<td>Q303=+1 ;MEAS. VALUE TRANSFER</td>
</tr>
</tbody>
</table>
15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)

Cycle run

Touch Probe Cycle 418 calculates the intersection of the lines connecting opposite holes and sets the datum at the intersection. If desired, the TNC can also enter the intersection into a datum table or preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to the center of the first hole 1.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole 2.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The TNC repeats steps 3 and 4 for holes 3 and 4.
6. Finally, the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482). 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:

**Danger of collision!**
If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
Touch Probe Cycles: Automatic Datum Setting

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

Cycle parameters

- **1st hole: Center in 1st axis** Q268 (absolute): Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st hole: Center in 2nd axis** Q269 (absolute): Center of the first hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 1st axis** Q270 (absolute): Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd hole: Center in 2nd axis** Q271 (absolute): Center of the second hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd center in 1st axis** Q316 (absolute): Center of the 3rd hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd center in 2nd axis** Q317 (absolute): Center of the 3rd hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th center in 1st axis** Q318 (absolute): Center of the 4th hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th center in 2nd axis** Q319 (absolute): Center of the 4th hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the line intersection. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0 input range -99999.9999 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>NC Blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 418 DATUM FROM 4 HOLES</td>
</tr>
<tr>
<td>Q268=+20 ; 1ST CENTER 1ST AXIS</td>
</tr>
<tr>
<td>Q269=+25 ; 1ST CENTER 2ND AXIS</td>
</tr>
<tr>
<td>Q270=+150; 2ND CENTER 1ST AXIS</td>
</tr>
<tr>
<td>Q271=+25 ; 2ND CENTER 2ND AXIS</td>
</tr>
<tr>
<td>Q316=+150; 3RD CENTER 1ST AXIS</td>
</tr>
<tr>
<td>Q317=+85 ; 3RD CENTER 2ND AXIS</td>
</tr>
<tr>
<td>Q318=+22 ; 4TH CENTER 1ST AXIS</td>
</tr>
<tr>
<td>Q319=+80 ; 4TH CENTER 2ND AXIS</td>
</tr>
<tr>
<td>Q261=−5 ; MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q260=+10 ; CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q305=12 ; NO. IN TABLE</td>
</tr>
<tr>
<td>Q331=+0 ; DATUM</td>
</tr>
<tr>
<td>Q332=+0 ; DATUM</td>
</tr>
<tr>
<td>Q303=+1 ; MEAS. VALUE TRANSFER</td>
</tr>
<tr>
<td>Q381=1 ; PROBE IN TS AXIS</td>
</tr>
<tr>
<td>Q382=+85 ; 1ST CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q383=+50 ; 2ND CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q384=+0 ; 3RD CO. FOR TS AXIS</td>
</tr>
<tr>
<td>Q333=+0 ; DATUM</td>
</tr>
</tbody>
</table>
New datum for minor axis Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0 input range -99999.9999 to 99999.9999

Measured-value transfer (0, 1) Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
-1: Do not use! Is entered by the TNC when old programs are read in (see “Characteristics common to all touch probe cycles for datum setting”, page 482)
0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).

Probe in TS axis Q381: Specify whether the TNC should also set the datum in the touch probe axis:
0: Do not set the datum in the touch probe axis
1: Set the datum in the touch probe axis

Probe TS axis: Coord. 1st axis Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1st input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 2nd axis Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

Probe TS axis: Coord. 3rd axis Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

New datum in TS axis Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

Cycle run

Touch Probe Cycle 419 measures any coordinate in any axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from $F_{MAX}$ column) (see "Executing touch probe cycles", page 454) to the programmed touch point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite the programmed probing direction.

2. Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.

3. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for datum setting", page 482).

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you want to save the datum of several axes in the preset table, you can use Cycle 419 several times in a row. However, you also have to reactivate the preset number after every run of Cycle 419. If you work with preset 0 as active preset, this process is not required.
Cycle parameters

- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Measuring axis (1...3: 1 = principal axis)** Q272:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis

### Axis assignment

<table>
<thead>
<tr>
<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>Z</td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>Y</td>
<td>Z</td>
<td>X</td>
</tr>
<tr>
<td>X</td>
<td>Y</td>
<td>Z</td>
</tr>
</tbody>
</table>

### NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 419 DATUM IN ONE AXIS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q263=+25 ; 1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264=+25 ; 1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q261=+25 ; MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ; SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+50 ; CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q272=+1 ; MEASURING AXIS</td>
</tr>
<tr>
<td>Q267=+1 ; TRAVERSE DIRECTION</td>
</tr>
<tr>
<td>Q305=0 ; NO. IN TABLE</td>
</tr>
<tr>
<td>Q333=+0 ; DATUM</td>
</tr>
<tr>
<td>Q303=+1 ; MEAS. VALUE TRANSFER</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Datum Setting

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

- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  - -1: Negative Traverse direction
  - +1: Positive traverse direction

- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface. If Q303=0: If you enter Q305=0, the TNC writes to line 0 of the datum table. Input range 0 to 99999

- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0
  - Input range -99999.9999 to 99999.9999

- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  - -1: Do not use! Is entered by the TNC when old programs are read in (see "Characteristics common to all touch probe cycles for datum setting", page 482)
  - 0: Write the measured datum into the active datum table. The reference system is the active workpiece coordinate system
  - 1: Write the measured datum into the preset table. The reference system is the machine coordinate system (REF system).
Example: Datum setting in center of a circular segment and on top surface of workpiece

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

```
0 BEGIN PGM CYC413 MM
1 TOOL CALL 69 Z
Call tool 0 to define the touch probe axis
2 TCH PROBE 413 DATUM OUTSIDE CIRCLE
   Q321=+25 ;CENTER 1ST AXIS
   Center of circle: X coordinate
   Q322=+25 ;CENTER 2ND AXIS
   Center of circle: Y coordinate
   Q262=30 ;NOMINAL DIAMETER
   Circle diameter
   Q325=+90 ;STARTING ANGLE
   Polar coordinate angle for 1st touch point
   Q247=+45 ;STEPPING ANGLE
   Stepping angle for calculating the starting points 2 to 4
   Q261=+5 ;MEASURING HEIGHT
   Coordinate in the touch probe axis in which the measurement is made
   Q320=2 ;SET-UP CLEARANCE
   Safety clearance in addition to SET_UP column
   Q260=+10 ;CLEARANCE HEIGHT
   Height in the touch probe axis at which the probe can traverse without collision
   Q301=0 ;MOVE TO CLEARANCE
   Do not move to clearance height between measuring points
   Q305=0 ;NO. IN TABLE
   Set display
   Q331=+0 ;DATUM
   Set the display in X to 0
   Q332=+10 ;DATUM
   Set the display in Y to 10
   Q303=+0 ;MEAS. VALUE TRANSFER
   Without function, since display is to be set
   Q381=1 ;PROBE IN TS AXIS
   Also set datum in the touch probe axis
   Q382=+25 ;1ST CO. FOR TS AXIS
   X coordinate of touch point
   Q383=+25 ;2ND CO. FOR TS AXIS
   Y coordinate of touch point
   Q384=+25 ;3RD CO. FOR TS AXIS
   Z coordinate of touch point
   Q333=+0 ;DATUM
   Set the display in Z to 0
   Q423=4 ;NO. OF PROBE POINTS
   Measure circle with 4 probes
   Q365=0 ;TYPE OF TRAVERSE
   Move on circular path between measuring points
3 CALL PGM 35K47
   Call part program
4 END PGM CYC413 MM
```
15.15 Example: Datum setting on top surface of workpiece and in center of a bolt hole circle

The measured bolt hole center shall be written in the preset table so that it may be used at a later time.

```
0 BEGIN PGM CYC416 MM
1 TOOL CALL 69 Z
2 TCH PROBE 417 DATUM IN TS AXIS
  Q263=+7.5 ;1ST POINT 1ST AXIS
  Touch point: X coordinate
  Q264=+7.5 ;1ST POINT 2ND AXIS
  Touch point: Y coordinate
  Q294=+25 ;1ST POINT 3RD AXIS
  Touch point: Z coordinate
  Q320=0 ;SET-UP CLEARANCE
  Safety clearance in addition to SET_UP column
  Q260=+50 ;CLEARANCE HEIGHT
  Height in the touch probe axis at which the probe can traverse without collision
  Q305=1 ;NO. IN TABLE
  Write Z coordinate in line 1
  Q333=+0 ;DATUM
  Set touch-probe axis to 0
  Q303=+1 ;MEAS. VALUE TRANSFER
  In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)
3 TCH PROBE 416 DATUM CIRCLE CENTER
  Q273=+35 ;CENTER IN 1ST AXIS
  Center of the bolt hole circle: X coordinate
  Q274=+35 ;CENTER IN 2ND AXIS
  Center of the bolt hole circle: Y coordinate
  Q262=50 ;NOMINAL DIAMETER
  Diameter of the bolt hole circle
  Q291=+90 ;ANGLE OF 1ST HOLE
  Polar coordinate angle for 1st hole center
  Q292=+180 ;ANGLE OF 2ND HOLE
  Polar coordinate angle for 2nd hole center
  Q293=+270 ;ANGLE OF 3RD HOLE
  Polar coordinate angle for 3rd hole center
  Q261=+15 ;MEASURING HEIGHT
  Coordinate in the touch probe axis in which the measurement is made
  Q260=+10 ;CLEARANCE HEIGHT
  Height in the touch probe axis at which the probe can traverse without collision
  Q305=1 ;NO. IN TABLE
  Enter center of bolt hole circle (X and Y) in line 1
  Q331=+0 ;DATUM
  Q332=+0 ;DATUM
```
Example: Datum setting on top surface of workpiece and in center 15.15 of a bolt hole circle

<table>
<thead>
<tr>
<th>Q303=+1 ;MEAS. VALUE TRANSFER</th>
<th>In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q381=0 ;PROBE IN TS AXIS</td>
<td>Do not set a datum in the touch probe axis</td>
</tr>
<tr>
<td>Q382=+0 ;1ST CO. FOR TS AXIS</td>
<td>No function</td>
</tr>
<tr>
<td>Q383=+0 ;2ND CO. FOR TS AXIS</td>
<td>No function</td>
</tr>
<tr>
<td>Q384=+0 ;3RD CO. FOR TS AXIS</td>
<td>No function</td>
</tr>
<tr>
<td>Q333=+0 ;DATUM</td>
<td>No function</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
<td>Safety clearance in addition to SET_UP column</td>
</tr>
<tr>
<td>4 CYCL DEF 247 DATUM SETTING</td>
<td>Activate new preset with Cycle 247</td>
</tr>
<tr>
<td>Q339=1 ;DATUM NUMBER</td>
<td>Call part program</td>
</tr>
<tr>
<td>6 CALL PGM 35KLZ</td>
<td></td>
</tr>
<tr>
<td>7 END PGM CYC416 MM</td>
<td></td>
</tr>
</tbody>
</table>
16

Touch Probe Cycles: Automatic Workpiece Inspection
16.1 Fundamentals

Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active. HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. Refer to your machine manual.

The TNC offers twelve cycles for measuring workpieces automatically.

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 REFERENCE PLANE</td>
<td>Measuring a coordinate in a selectable axis</td>
<td>540</td>
</tr>
<tr>
<td>1 POLAR DATUM PLANE</td>
<td>Measuring a point in a probing direction</td>
<td>541</td>
</tr>
<tr>
<td>420 MEASURE ANGLE</td>
<td>Measuring an angle in the working plane</td>
<td>542</td>
</tr>
<tr>
<td>421 MEASURE HOLE</td>
<td>Measuring the position and diameter of a hole</td>
<td>545</td>
</tr>
<tr>
<td>422 MEASURE CIRCLE OUTSIDE</td>
<td>Measuring the position and diameter of a circular stud</td>
<td>550</td>
</tr>
<tr>
<td>423 MEASURE RECTANGLE INSIDE</td>
<td>Measuring the position, length and width of a rectangular pocket</td>
<td>555</td>
</tr>
<tr>
<td>424 MEASURE RECTANGLE OUTSIDE</td>
<td>Measuring the position, length and width of a rectangular stud</td>
<td>558</td>
</tr>
<tr>
<td>425 MEASURE INSIDE WIDTH (2nd soft-key level)</td>
<td>Measuring slot width</td>
<td>561</td>
</tr>
<tr>
<td>426 MEASURE RIDGE WIDTH (2nd soft-key row)</td>
<td>Measuring the width of a ridge</td>
<td>564</td>
</tr>
</tbody>
</table>
**Fundamentals 16.1**

### Soft key | Cycle | Page
--- | --- | ---
| | | |
| | 427 MEASURE COORDINATE | 567
(2nd soft-key row) Measuring any coordinate in a selectable axis
| | 430 MEASURE BOLT HOLE CIRCLE | 570
(2nd soft-key row) Measuring position and diameter of a bolt hole circle
| | 431 MEASURE PLANE | 573
(2nd soft-key row) Measuring the A and B axis angles of a plane

**Recording the results of measurement**

For all cycles in which you automatically measure workpieces (with the exception of Cycles 0 and 1), you can have the TNC record the measurement results. In the respective probing cycle you can define if the TNC is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file the TNC, by default, saves the data as an ASCII file. The TNC will save the file in the directory that also contains the associated NC program.

Use the HEIDENHAIN data transfer software TNCRemo if you wish to output the measuring log over the data interface.
Example: Measuring log for touch probe cycle 421:

**Measuring log for Probing Cycle 421 Hole Measuring**

Date: 30-06-2005  
Time: 6:55:04  
Measuring program: TNC:\GEH35712\CHECK1.H

**Nominal values:**  
Center in reference axis: 50.0000  
Center in minor axis: 65.0000  
Diameter: 12.0000

**Given limit values:**  
Maximum limit for center in reference axis: 50.1000  
Minimum limit for center in reference axis: 49.9000  
Maximum limit for center in minor axis: 65.1000  
Minimum limit for center in minor axis: 64.9000  
Maximum dimension for hole: 12.0450  
Minimum dimension for hole: 12.0000

**Actual values:**  
Center in reference axis: 50.0810  
Center in minor axis: 64.9530  
Diameter: 12.0259

**Deviations:**  
Center in reference axis: 0.0810  
Center in minor axis: -0.0470  
Diameter: 0.0259

**Further measuring results:** Measuring height: -5.0000

**End of measuring log**
Measurement results in Q parameters
The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. Deviations from the nominal value are saved in the parameters Q161 to Q166. Note the table of result parameters listed with every cycle description.
During cycle definition the TNC also shows the result parameters for the respective cycle in a help graphic (see figure at upper right). The highlighted result parameter belongs to that input parameter.

Classification of results
For some cycles you can inquire the status of measuring results through the globally effective Q parameters Q180 to Q182.

<table>
<thead>
<tr>
<th>Class of results</th>
<th>Parameter value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measurement results are within</td>
<td>Q180 = 1</td>
</tr>
<tr>
<td>tolerance</td>
<td></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.

Milling tool: If you reference parameter Q330 to a milling tool, the appropriate values are compensated in the following way: the TNC basically always compensates the tool radius in column DR of the tool table, even if the measured deviation is within the stated tolerance. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: must be reworked).

Turning tool: (applies to cycles 421, 422, 427 only) If you reference parameter Q330 to a turning tool, the appropriate values in row DZL and DXL, respectively, are compensated. The TNC monitors the breakage tolerance which is defined in column LBREAK. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: must be reworked).
**Tool breakage monitoring**

This function works only:
- If the tool table is active.
- If tool monitoring is switched on in the cycle (enter Q330 not equal to 0).
- If the breakage tolerance RBREAK for the tool number entered in the table is greater than 0 (see also the User’s Manual, section 5.2 "Tool Data").

The TNC will output an error message and stop program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time the tool will be deactivated in the tool table (column TL = L).

**Reference system for measurement results**

The TNC transfers all the measurement results to the result parameters and the log file in the active coordinate system, or as the case may be, the shifted and/or rotated/tilted coordinate system.
16.2 DATUM PLANE (Cycle 0, DIN/ISO: G55)

Cycle run
1. The touch probe moves at rapid traverse (value from FMAX column) to the starting position programmed in the cycle.
2. Then the touch probe runs the probing process at the probing feed rate (column F). The probing direction is defined in the cycle.
3. After the TNC has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in the parameters Q115 to Q119. For the values in these parameters the TNC does not account for the stylus length and radius.

Please note while programming:

Danger of collision!
Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

Cycle parameters

- **Parameter number for result**: Enter the number of the Q parameter to which you want to assign the coordinate. Input range 0 to 1999
- **Probing axis/Probing direction**: Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Confirm your entry with the ENT key. Input range: All NC axes
- **Nominal position value**: Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- **To conclude the input, press the ENT key.**

NC blocks

67 TCH PROBE 0.0 REF. PLANE Q5 X-
68 TCH PROBE 0.1 X+5 Y+0 Z-5
16.3 POLAR DATUM PLANE (Cycle 1)

Cycle run
Touch Probe Cycle 1 measures any position on the workpiece in any direction.
1. The touch probe moves at rapid traverse (value from FMAX column) to the starting position programmed in the cycle.
2. Then the touch probe runs the probing process at the probing feed rate (column F). During probing the TNC moves simultaneously in two axes (depending on the probing angle). The probing direction is defined by the polar angle entered in the cycle.
3. After the TNC has saved the position, the probe returns to the starting point. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in parameters Q115 to Q119.

Please note while programming:

Danger of collision!
Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

The probing axis defined in the cycle specifies the probing plane:
- Probing axis X: X/Y plane
- Probing axis Y: Y/Z plane
- Probing axis Z: Z/X plane

Cycle parameters
- **Probing axis:** Enter the probing axis with the axis selection keys or ASCII keyboard. Confirm your entry with the ENT key. Input range: X, Y or Z
- **Probing angle:** Angle, measured from the probing axis, at which the touch probe is to move. Input range -180.0000 to 180.0000
- **Nominal position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- To conclude the input, press the ENT key.

NC blocks
- 67 TCH PROBE 1.0 POLAR REFERENCE PLANE
- 68 TCH PROBE 1.1 X ANGLE: +30
- 69 TCH PROBE 1.2 X+5 Y+0 Z-5
**16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)**

**Cycle run**

Touch Probe Cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \( \text{FMAX} \) column) (see "Executing touch probe cycles", page 454) to the programmed touch point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \( \text{F} \)).

3. Then the touch probe moves to the next starting position 2 and from there probes the second position.

4. The TNC returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

<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
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points

NC blocks

```
TCH PROBE 420 MEASURE ANGLE
Q263=+10 ;1ST POINT 1ST AXIS
Q264=+10 ;1ST POINT 2ND AXIS
Q265=+15 ;2ND POINT 1ST AXIS
Q266=+95 ;2ND POINT 2ND AXIS
Q272=1 ;MEASURING AXIS
Q267=-1 ;TRAVERSE DIRECTION
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SET-UP CLEARANCE
Q260=+10 ;CLEARANCE HEIGHT
Q301=1 ;MOVE TO CLEARANCE
Q281=1 ;MEASURING LOG
```
16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file **TCHPR420.TXT** as standard in the directory **TNC\**.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)

Cycle run

Touch Probe Cycle 421 measures the center and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1 Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.

3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4 The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

<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°

If you reference parameter Q330 to a turning tool, the following applies:
- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the TNC compensates the turning tool, the appropriate values in line DZL and DXL, respectively, are compensated.
- The TNC monitors the breakage tolerance which is defined in column LBREAK

If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect.
Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute value): Center of the hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the diameter of the hole. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Maximum limit of size for hole** Q275: Maximum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- **Minimum limit of size for hole** Q276: Minimum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>NC block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<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>Q262=75</td>
<td>NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q325=+0</td>
<td>STARTING ANGLE</td>
</tr>
<tr>
<td>Q247=+60</td>
<td>STEPPING ANGLE</td>
</tr>
<tr>
<td>Q261=-5</td>
<td>MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0</td>
<td>SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20</td>
<td>CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=1</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q275=75.12</td>
<td>MAXIMUM DIMENSION</td>
</tr>
<tr>
<td>Q276=74.95</td>
<td>MINIMUM DIMENSION</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=0</td>
<td>TOOL</td>
</tr>
<tr>
<td>Q423=4</td>
<td>NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q365=1</td>
<td>TYPE OF TRAVERSE</td>
</tr>
<tr>
<td>Q498=0</td>
<td>REVERSE TOOL</td>
</tr>
<tr>
<td>Q531=0</td>
<td>INCLINED ANGLE</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Workpiece Inspection

16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)

- **Measuring log** Q281: Definition of whether TNC should create a measuring log:
  0: Create no measuring log
  1: Create measuring log. The TNC will save the log file TCHPR421.TXT by default in the directory that also contains the associated NC program.
  2: Interrupt the program run and display the measuring log on the TNC screen. Resume program run with NC Start.

- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message

- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 538). Enter 0 to 32767, optionally the tool name with up to 16 characters
  0: Monitoring not active
  >0: Number or name of the tool the TNC used to perform the operation with. You are able to apply a tool via soft key directly from the tool table.

- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points

- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations
Reverse tool (0=no/1=yes)? Q498: Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the TNC requires the exact working condition. Therefore, enter the following:
- 1: Turning tool is mirrored (rotated by 180°) e.g. by Cycle 800 and parameter Reverse the tool Q498=1
- 0: Turning tool matches the description of the turning tool table totoolturn.trn, no modifications, e.g. via Cycle 800 and parameter Reverse the tool Q498=0

Angle of incidence? Q531: Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence between the turning tool and the work piece during machining, e.g. from Cycle 800 parameter Angle of incidence? Q531. Input range: -180° to +180°
16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

**Cycle run**

Touch Probe Cycle 422 measures the center and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \(F_{MAX}\) column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the \(SET\_UP\) column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \(F\)). The TNC derives the probing direction automatically from the programmed starting angle.

3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

<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°

If you reference parameter Q330 to a turning tool, the following applies:
- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the TNC compensates the turning tool, the appropriate values in line DZL and DXL, respectively, are compensated.
- The TNC monitors the breakage tolerance which is defined in column LBREAK

If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect.
Touch Probe Cycles: Automatic Workpiece Inspection

16.6 MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422)

Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the diameter of the stud. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Maximum limit of size for stud** Q277: Maximum permissible diameter for the stud. Input range 0 to 99999.9999
- **Minimum limit of size for the stud** Q278: Minimum permissible diameter for the stud. Input range 0 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 422 MEAS. CIRCLE OUTSIDE</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>Q262=75 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q325=+90 ;STARTING ANGLE</td>
</tr>
<tr>
<td>Q247=+30 ;STEPPING ANGLE</td>
</tr>
<tr>
<td>Q261=5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+10 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q275=35.15 ;MAXIMUM DIMENSION</td>
</tr>
<tr>
<td>Q276=34.9 ;MINIMUM DIMENSION</td>
</tr>
<tr>
<td>Q279=0.05 ;TOLERANCE 1ST CENTER</td>
</tr>
</tbody>
</table>
MEASURE HOLE OUTSIDE (Cycle 422, DIN/ISO: G422) 16.6

- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file TCHPR422.TXT as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 538). Input range 0 to 32767.9, alternatively tool name with up to 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" (Q301=1) is active:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

| Q280=0.05 ; TOLERANCE 2ND CENTER |
| Q281=1 ; MEASURING LOG |
| Q309=0 ; PGM STOP IF ERROR |
| Q330=0 ; TOOL |
| Q423=4 ; NO. OF PROBE POINTS |
| Q365=1 ; TYPE OF TRAVERSE |
| Q498=0 ; REVERSE TOOL |
| Q531=0 ; INCLINED ANGLE |
Reverse tool (0=no/1=yes)? Q498: Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the TNC requires the exact working condition. Therefore, enter the following:
- **1**: Turning tool is mirrored (rotated by 180°) e.g. by Cycle 800 and parameter Reverse the tool Q498=1
- **0**: Turning tool matches the description of the turning tool table toolturn.trn, no modifications, e.g. via Cycle 800 and parameter Reverse the tool Q498=0

Angle of incidence? Q531: Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence between the turning tool and the work piece during machining, e.g. from Cycle 800 parameter Angle of incidence? Q531. Input range: -180° to +180°
Cycle run

Touch Probe Cycle 423 finds the center, length and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \( F_{MAX} \) column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the \( SET_UP \) column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \( F \)).

3. Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following \( Q \) parameters:

<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.
Touch Probe Cycles: Automatic Workpiece Inspection

16.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423)

Cycle parameters

- **Center in 1st axis** Q273 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q282: Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q283: Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points
- **Max. size limit 1st side length** Q284: Maximum permissible length of the pocket. Input range 0 to 99999.9999
- **Min. size limit 1st side length** Q285: Minimum permissible length of the pocket. Input range 0 to 99999.9999
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the pocket. Input range 0 to 99999.9999
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the pocket. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 423 MEAS. RECTAN. INSIDE</td>
</tr>
<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 ;SET-UP 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=0 ;TOOL</td>
</tr>
</tbody>
</table>
Measuring log  Q281: Define whether the TNC should create a measuring log:
0: Do not create a measuring log
1: Create a measuring log: The TNC saves the log file TCHPR423.TXT as standard in the directory TNC:
2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.

PGM stop if tolerance error  Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
0: Do not interrupt program run, do not output an error message
1: Interrupt program run and output an error message

Tool for monitoring  Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring“, page 538). Input range 0 to 32767, alternatively tool name with up to 16 characters
0: Monitoring inactive
> 0: Tool number in the tool table TOOL.T
16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)

Cycle run

Touch Probe Cycle 424 finds the center, length and width of a rectangular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F).

3. Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

<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
- **1st side length** Q282: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q283: Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
- **Max. size limit 1st side length** Q284: Maximum permissible length of the stud. Input range 0 to 99999.9999
- **Min. size limit 1st side length** Q285: Minimum permissible length of the stud. Input range 0 to 99999.9999
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the stud. Input range 0 to 99999.9999
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the stud. Input range 0 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 424 MEAS. RECTAN. OUTS.</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=75 ;FIRST SIDE LENGTH</td>
</tr>
<tr>
<td>Q283=35 ;2ND SIDE LENGTH</td>
</tr>
<tr>
<td>Q261=5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q284=75.1 ;MAX. LIMIT 1ST SIDE</td>
</tr>
<tr>
<td>Q285=74.9 ;MIN. LIMIT 1ST SIDE</td>
</tr>
<tr>
<td>Q286=35 ;MAX. LIMIT 2ND SIDE</td>
</tr>
<tr>
<td>Q287=34.95 ;MIN. LIMIT 2ND SIDE</td>
</tr>
</tbody>
</table>
16.8  MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)

- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file TCHPR424.TXT as standard in the directory TNC:
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 538). Input range 0 to 32767.9, alternatively tool name with maximum of 16 characters
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T

| Q279 = 0.1 | ;TOLERANCE 1ST CENTER |
| Q280 = 0.1 | ;TOLERANCE 2ND CENTER |
| Q281 = 1 | ;MEASURING LOG |
| Q309 = 0 | ;PGM STOP IF ERROR |
| Q330 = 0 | ;TOOL |
16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)

Cycle run

Touch Probe Cycle 425 measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in a system parameter.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The first probing is always in the positive direction of the programmed axis.

3. If you enter an offset for the second measurement, the TNC then moves the touch probe (if required, at clearance height) to the next starting point 2 and probes the second touch point. If the nominal length is large, the TNC moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the TNC measures the width in the exact opposite direction.

4. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation value in the following Q parameters:

<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: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Nominal length** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999
- **Maximum dimension** Q288: Maximum permissible length. Input range 0 to 99999.9999
- **Minimum dimension** Q289: Minimum permissible length. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file TCHPR425.TXT as standard in the directory TNCA.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message

### NC blocks

<table>
<thead>
<tr>
<th>NC Blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q328=+75 ;STARTNG PNT 1ST AXIS</td>
</tr>
<tr>
<td>Q329=-12.5;STARTNG PNT 2ND AXIS</td>
</tr>
<tr>
<td>Q310=+0 ;OFFS. 2ND MEASUREMENT</td>
</tr>
<tr>
<td>Q272=1 ;MEASURING 2ND AXN</td>
</tr>
<tr>
<td>Q261=+5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q260=+10 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q311=+25 ;NOMINAL LENGTH</td>
</tr>
<tr>
<td>Q288=+25.05;MAXIMUM DIMENSION</td>
</tr>
<tr>
<td>Q289=+25 ;MINIMUM DIMENSION</td>
</tr>
<tr>
<td>Q281=1 ;MEASURING LOG</td>
</tr>
<tr>
<td>Q309=0 ;PGM STOP IF ERROR</td>
</tr>
<tr>
<td>Q330=0 ;TOOL</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
</tr>
</tbody>
</table>
Tool for monitoring Q330: Definition of whether the TNC is to monitor the tool (see “Tool monitoring”, page 538). Enter 0 to 32767.9, optionally the tool name with up to 16 characters. 0: Monitoring not active
>0: Number or name of the tool the TNC used to perform the operation with. You are able to apply a tool via soft key directly from the tool table.

Set-up clearance Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table), and is only effective when the datum is probed in the touch probe axis. Input range 0 to 99999.9999

Traversing to clearance height Q301: definition of how the touch probe is to move between the measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Touch Probe Cycles: Automatic Workpiece Inspection

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

**Cycle run**

Touch Probe Cycle 426 measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from \(F_{\text{MAX}}\) column) (see "Executing touch probe cycles", page 454) to touch point 1. The TNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column \(F\)). The first probing is always in the negative direction of the programmed axis.

3. Then the touch probe moves at clearance height to the next starting position and probes the second touch point.

4. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation value in the following Q parameters:

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

Cycle parameters

- **1st meas. point 1st axis Q263 (absolute):** Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis Q264 (absolute):** Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis Q265 (absolute):** Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis Q266 (absolute):** Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis Q272:** Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Measuring height in the touch probe axis Q261 (absolute):** Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance Q320 (incremental):** Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Clearance height Q260 (absolute):** Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Nominal length Q311:** Nominal value of the length to be measured. Input range 0 to 99999.9999
- **Maximum dimension Q288:** Maximum permissible length. Input range 0 to 99999.9999
- **Minimum dimension Q289:** Minimum permissible length. Input range 0 to 99999.9999
- **Measuring log Q281:** Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file TCHPR426.TXT as standard in the directory TNC:
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 426 MEASURE RIDGE WIDTH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q263==+50 ;1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264==+25 ;1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q265==+50 ;2ND POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q266==+85 ;2ND POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q272=2 ;MEASURING AXIS</td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q311=45 ;NOMINAL LENGTH</td>
</tr>
<tr>
<td>Q288=+45 ;MAXIMUM DIMENSION</td>
</tr>
<tr>
<td>Q289=-44.95 ;MINIMUM DIMENSION</td>
</tr>
<tr>
<td>Q281=1 ;MEASURING LOG</td>
</tr>
<tr>
<td>Q309=0 ;PGM STOP IF ERROR</td>
</tr>
<tr>
<td>Q330=0 ;TOOL</td>
</tr>
</tbody>
</table>
16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)

- **PGM stop if tolerance error Q309**: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  - 0: Do not interrupt program run, do not output an error message
  - 1: Interrupt program run and output an error message

- **Tool for monitoring Q330**: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 538). Input range 0 to 32767, alternatively tool name with up to 16 characters
  - 0: Monitoring inactive
  - > 0: Tool number in the tool table TOOL.T
16.11  MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)

Cycle run

Touch Probe Cycle 427 finds a coordinate in a selectable axis and saves the value in a system parameter. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to touch point 1.
   The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2. Then the TNC positions the touch probe to the entered touch point 1 in the working plane and measures the actual value in the selected axis.
3. Finally the TNC returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter.

<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.

If an axis of the active working plane is defined as measuring axis (Q272 = 1 or 2), the TNC compensates the tool radius. From the defined traversing direction (Q267) the TNC determines the direction of compensation.

If the touch probe axis is defined as measuring axis (Q272 = 3), the TNC compensates the tool length.

If you reference parameter Q330 to a turning tool, the following applies:
- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the TNC compensates the turning tool, the appropriate values in line DZL and DXL, respectively, are compensated.
- The TNC monitors the breakage tolerance which is defined in column LBREAK

If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect.
**Touch Probe Cycles: Automatic Workpiece Inspection**

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

#### Cycle parameters

- **1st meas. point 1st axis Q263** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis Q264** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis Q261** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance Q320** (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **Measuring axis (1 to 3: 1 = Reference axis) Q272**: Axis in which the measurement is to be made:
  1. Reference axis = measuring axis
  2. Minor axis = measuring axis
  3. Touch probe axis = measuring axis
- **Traverse direction 1 Q267**: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction
- **Clearance height Q260** (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Measuring log Q281**: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log. The TNC saves the log file TCHPR427.TXT as standard in the directory TNC:
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.

#### NC blocks

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 427 MEASURE COORDINATE</td>
</tr>
<tr>
<td>Q263=+35 ;1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264=+45 ;1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q261=+5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q272=3 ;MEASURING AXIS</td>
</tr>
<tr>
<td>Q267=1 ;TRAVERSE DIRECTION</td>
</tr>
<tr>
<td>Q260=+20 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q281=1 ;MEASURING LOG</td>
</tr>
<tr>
<td>Q288=5.1 ;MAXIMUM DIMENSION</td>
</tr>
<tr>
<td>Q289=4.95 ;MINIMUM DIMENSION</td>
</tr>
</tbody>
</table>
Maximum limit of size Q288: Maximum permissible measured value. Input range 0 to 99999.9999

Minimum limit of size Q289: Minimum permissible measured value. Input range 0 to 99999.9999

PGM stop if tolerance error Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
0: Do not interrupt program run, do not output an error message
1: Interrupt program run and output an error message

Tool for monitoring Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring", page 538). Input range 0 to 32767.9, alternatively tool name with up to 16 characters
0: Monitoring inactive
> 0: Tool number in the tool table TOOL.T

Reverse tool (0=no/1=yes)? Q498: Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the TNC requires the exact working condition. Therefore, enter the following:
1: Turning tool is mirrored (rotated by 180°) e.g. by Cycle 800 and parameter Reverse the tool Q498=1
0: Turning tool matches the description of the turning tool table toolturn.trn, no modifications, e.g. via Cycle 800 and parameter Reverse the tool Q498=0

Angle of incidence? Q531: Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence between the turning tool and the work piece during machining, e.g. from Cycle 800 parameter Angle of incidence? Q531. Input range: -180° to +180°
16.12 MEASURE BOLT HOLE CIRCLE
(Cycle 430, DIN/ISO: G430)

Cycle run
Touch Probe Cycle 430 finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from $F_{\text{MAX}}$ column) (see "Executing touch probe cycles", page 454) to the center of the first hole 1.

2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.

3. The touch probe returns to the clearance height and then to the position entered as center of the second hole 2.

4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.

5. The touch probe returns to the clearance height and then to the position entered as center of the third hole 3.

6. The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.

7. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

<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; there is no automatic tool compensation.

Cycle parameters

- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the bolt hole circle diameter. Input range 0 to 99999.9999
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999
- **Maximum limit of size** Q288: Maximum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- **Minimum limit of size** Q289: Minimum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999

NC blocks

<table>
<thead>
<tr>
<th>5 TCH PROBE 430 MEAS. BOLT HOLE CIRC</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>Q262=80 ;NOMINAL DIAMETER</td>
</tr>
<tr>
<td>Q291=+0 ;ANGLE OF 1ST HOLE</td>
</tr>
<tr>
<td>Q292=+90 ;ANGLE OF 2ND HOLE</td>
</tr>
<tr>
<td>Q293=+180;ANGLE OF 3RD HOLE</td>
</tr>
<tr>
<td>Q261=-5 ;MEASURING HEIGHT</td>
</tr>
<tr>
<td>Q260=-10 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q288=80.1 ;MAXIMUM DIMENSION</td>
</tr>
<tr>
<td>Q289=79.9 ;MINIMUM DIMENSION</td>
</tr>
<tr>
<td>Q279=0.15 ;TOLERANCE 1ST CENTER</td>
</tr>
</tbody>
</table>
16

Touch Probe Cycles: Automatic Workpiece Inspection

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

- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file TCHPR430.TXT as standard in the directory TNC:\.
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0: Do not interrupt program run, do not output an error message
  1: Interrupt program run and output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor for tool breakage (see "Tool monitoring", page 538). Input range 0 to 327679, alternatively tool name with up to 16 characters.
  0: Monitoring inactive
  > 0: Tool number in the tool table TOOL.T
16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)

Cycle run

Touch Probe Cycle 431 finds the angle of a plane by measuring three points. It saves the measured values in system parameters.

1. Following the positioning logic, the TNC positions the touch probe at rapid traverse (value from FMAX column) (see "Executing touch probe cycles", page 454) to the programmed touch point 1 and measures the first point of the plane. The TNC offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.

2. The touch probe returns to the clearance height and then moves in the working plane to starting point 2 and measures the actual value of the second touch point of the plane.

3. The touch probe returns to the clearance height and then moves in the working plane to starting point 3 and measures the actual value of the third touch point of the plane.

4. Finally the TNC returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

<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 (first to third measurement)</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Workpiece Inspection

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

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
For the TNC to be able to calculate the angular values, the three measuring points must not be positioned on one straight line.
The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172.
With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.
The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

Cycle parameters

- **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
MEASURE PLANE (Cycle 431, DIN/ISO: G431) 16.13

- **3rd meas. point 1st axis** Q296 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd meas. point 2nd axis** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999

- **3rd meas. point 3rd axis** Q298 (absolute): Coordinate of the third touch point in the touch probe axis. Input range -99999.9999 to 99999.9999

- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999

- **Measuring log** Q281: Define whether the TNC should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The TNC saves the log file TCHPR431.TXT as standard in the directory TNC:
  2: Interrupt program run and output measuring log to the TNC screen. Resume program run with NC Start.

---

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 431 MEASURE PLANE</td>
</tr>
<tr>
<td>Q263=+20 ;1ST POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q264=+20 ;1ST POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q294=-10 ;1ST POINT 3RD AXIS</td>
</tr>
<tr>
<td>Q265=+50 ;2ND POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q266=+80 ;2ND POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q295=+0  ;2ND POINT 3RD AXIS</td>
</tr>
<tr>
<td>Q296=+90 ;3RD POINT 1ST AXIS</td>
</tr>
<tr>
<td>Q297=+35 ;3RD POINT 2ND AXIS</td>
</tr>
<tr>
<td>Q298=+12 ;3RD POINT 3RD AXIS</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q260=+5 ;CLEARANCE HEIGHT</td>
</tr>
<tr>
<td>Q281=1 ;MEASURING LOG</td>
</tr>
</tbody>
</table>
### 16.14 Programming Examples

**Example: Measuring and reworking a rectangular stud**

**Program sequence**
- Roughing with 0.5 mm finishing allowance
- Measuring
- Rectangular stud finishing in accordance with the measured values

```plaintext
<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 B</td>
<td>BEGIN PGM BEAMS MM</td>
<td></td>
</tr>
<tr>
<td>1 T</td>
<td>TOOL CALL 69 Z</td>
<td>Tool call for roughing</td>
</tr>
<tr>
<td>2 L</td>
<td>Z+100 R0 FMAX</td>
<td>Retract the tool</td>
</tr>
<tr>
<td>3 F</td>
<td>N0: Q1 = +81</td>
<td>Length of rectangle in X (roughing dimension)</td>
</tr>
<tr>
<td>4 F</td>
<td>N0: Q2 = +61</td>
<td>Length of rectangle in Y (roughing dimension)</td>
</tr>
<tr>
<td>5 C</td>
<td>CALL LBL 1</td>
<td>Call subprogram for machining</td>
</tr>
<tr>
<td>6 L</td>
<td>Z+100 R0 FMAX</td>
<td>Retract the tool, change the tool</td>
</tr>
<tr>
<td>7 T</td>
<td>TOOL CALL 99 Z</td>
<td>Call the touch probe</td>
</tr>
<tr>
<td>8 T</td>
<td>CH PROBE 424 MEAS. RECTAN. OUTS.</td>
<td>Measure the rough-milled rectangle</td>
</tr>
<tr>
<td></td>
<td>Q273=+50 ;CENTER IN 1ST AXIS</td>
<td>Nominal length in X (final dimension)</td>
</tr>
<tr>
<td></td>
<td>Q274=+50 ;CENTER IN 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q282=80 ;FIRST SIDE LENGTH</td>
<td>Nominal length in Y (final dimension)</td>
</tr>
<tr>
<td></td>
<td>Q283=60 ;2ND SIDE LENGTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q261=-5 ;MEASURING HEIGHT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q320=0 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q260=+30 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q301=0 ;MOVE TO CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q284=0 ;MAX. LIMIT 1ST SIDE</td>
<td>Input values for tolerance checking not required</td>
</tr>
<tr>
<td></td>
<td>Q285=0 ;MIN. LIMIT 1ST SIDE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q286=0 ;MAX. LIMIT 2ND SIDE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q287=0 ;MIN. LIMIT 2ND SIDE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q279=0 ;TOLERANCE 1ST CENTER</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q280=0 ;TOLERANCE 2ND CENTER</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q281=0 ;MEASURING LOG</td>
<td>No measuring log transmission</td>
</tr>
<tr>
<td></td>
<td>Q309=0 ;PGM STOP IF ERROR</td>
<td>Do not output an error message</td>
</tr>
<tr>
<td></td>
<td>Q330=0 ;TOOL NO.</td>
<td>No tool monitoring</td>
</tr>
<tr>
<td>9 F</td>
<td>N2: Q1 = +Q1 - +Q164</td>
<td>Calculate length in X including the measured deviation</td>
</tr>
<tr>
<td>10 F</td>
<td>N2: Q2 = +Q2 - +Q165</td>
<td>Calculate length in Y including the measured deviation</td>
</tr>
<tr>
<td>11 L</td>
<td>Z+100 R0 FMAX</td>
<td>Retract the touch probe, change the tool</td>
</tr>
</tbody>
</table>
```
### Programming Examples 16.14

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>TOOL CALL 1 Z 55000</td>
<td>Tool call for finishing</td>
</tr>
<tr>
<td>13</td>
<td>CALL LBL 1</td>
<td>Call subprogram for machining</td>
</tr>
<tr>
<td>14</td>
<td>L Z+100 R0 FMAX M2</td>
<td>Retract in the tool axis, end program</td>
</tr>
<tr>
<td>15</td>
<td>LBL 1</td>
<td>Subprogram with fixed cycle for rectangular stud</td>
</tr>
<tr>
<td>16</td>
<td>CYCL DEF 213 STUD FINISHING</td>
<td>Subprogram with fixed cycle for rectangular stud</td>
</tr>
<tr>
<td></td>
<td>Q200=20 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q201=-10 ;DEPTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q206=150 ;FEED RATE FOR PLUNGNG</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q202=5 ;PLUNGING DEPTH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q207=500 ;FEED RATE FOR MILLING</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q203=+10 ;SURFACE COORDINATE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q204=20 ;2ND SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q216=+50 ;CENTER IN 1ST AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q217=+50 ;CENTER IN 2ND AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q218=Q1 ;FIRST SIDE LENGTH</td>
<td>Length in X variable for roughing and finishing</td>
</tr>
<tr>
<td></td>
<td>Q219=Q2 ;SECOND SIDE LENGTH</td>
<td>Length in Y variable for roughing and finishing</td>
</tr>
<tr>
<td></td>
<td>Q220=0 ;CORNER RADIUS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q221=0 ;ALLOWANCE IN 1ST AXS</td>
<td></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>
**Example: Measuring a rectangular pocket and recording the results**

```
0 BEGIN PGM BSMEAS MM
1 TOOL CALL 1 Z  
   Tool call for touch probe
2 L Z+100 R0 FMAX  
   Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE
   Q273=+50 ;CENTER IN 1ST AXIS
   Q274=+40 ;CENTER IN 2ND AXIS
   Q282=90 ;FIRST SIDE LENGTH  
               Nominal length in X
   Q283=70 ;2ND SIDE LENGTH  
               Nominal length in Y
   Q261=-5 ;MEASURING HEIGHT
   Q260=+20 ;CLEARANCE HEIGHT
   Q301=0 ;MOVE TO CLEARANCE
   Q284=90.15 ;MAX. LIMIT 1ST SIDE  
               Maximum limit in X
   Q285=89.95 ;MIN. LIMIT 1ST SIDE  
               Minimum limit in X
   Q286=70.1 ;MAX. LIMIT 2ND SIDE  
               Maximum limit in Y
   Q287=69.9 ;MIN. LIMIT 2ND SIDE  
               Minimum limit in Y
   Q279=0.15 ;TOLERANCE 1ST CENTER  
               Permissible position deviation in X
   Q280=0.1 ;TOLERANCE 2ND CENTER  
               Permissible position deviation in Y
   Q281=1 ;MEASURING LOG  
               Save measuring log to a file
   Q309=0 ;PGM STOP IF ERROR  
               Do not display an error message in case of a tolerance violation
   Q330=0 ;TOOL NUMBER  
               No tool monitoring
4 L Z+100 R0 FMAX M2  
   Retract the tool, end program
5 END PGM BSMEAS MM
```
17

Touch Probe Cycles: Special Functions
17.1 Fundamentals

Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used.

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

The TNC provides a cycle for the following special purpose:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3 MEASURING</td>
<td>581</td>
</tr>
<tr>
<td></td>
<td>Cycle for defining OEM cycles</td>
<td></td>
</tr>
</tbody>
</table>
17.2 MEASURE (Cycle 3)

Cycle run

Touch Probe Cycle 3 measures any position on the workpiece in a selectable direction. Unlike other measuring cycles, Cycle 3 enables you to enter the measuring range \textit{SET UP} and feed rate \textit{F} directly. Also, the touch probe retracts by a definable value after determining the measured value \textit{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 \textit{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 \textit{DIST} (maximum traverse to touch point) and \textit{F} (probing feed rate) data from the touch-probe table, which are effective in other measuring cycles, do not apply in Touch Probe Cycle 3.

Remember that the TNC always writes to four successive Q-parameters.

If the TNC was not able to determine a valid touch point, the program is run without error message. In this case the TNC assigns the value \(-1\) to the 4th result parameter so that you can deal with the error yourself.

The TNC retracts the touch probe by no more than the retraction distance \textit{MB} and does not pass the starting point of the measurement. This rules out any collision during retraction.

With function \textbf{FN17: SYSWRITE ID 990 NR 6} you can set whether the cycle runs through the probe input X12 or X13.
**Touch Probe Cycles: Special Functions**

### 17.2  MEASURE (Cycle 3)

**Cycle parameters**

- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters. Input range: 0 to 1999

- **Probing axis**: Enter the axis in whose direction the probe is to move and confirm with the ENT key. Input range: X, Y or Z

- **Probing angle**: Angle, measured from the defined probing axis in which the touch probe is to move. Confirm with ENT. Input range –180.0000 to 180.0000

- **Maximum measuring range**: Enter the maximum distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range -99999.9999 to 99999.9999

- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000

- **Maximum retraction distance**: Traverse path in the direction opposite the probing direction, after the stylus was deflected. The TNC returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input range 0 to 99999.9999

- **Reference system? (0=ACTUAL/1=REF)**: Define whether the probing direction and measuring result should reference the current coordinate system (ACTUAL, can be shifted or rotated) or the machine coordinate system (REF):
  - 0: Probe in the current system and save the measuring result to the ACTUAL system
  - 1: Probe in the fixed machine REF system and save the measuring result to the REF system.

- **Error mode (0=OFF/1=ON)**: Specify whether the TNC is to issue an error message if the stylus is deflected at cycle start. If mode 1 is selected, the TNC saves the value -1 in the 4th result parameter and continues the cycle:
  - 0: Error message output
  - 1: No error message output

**NC blocks**

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</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>
17.3 MEASURING IN 3-D (Cycle 4)

Cycle run

Cycle 4 is an auxiliary cycle that can be used for probing with any touch probe (TS, TT or TL). The TNC does not provide a cycle for calibrating the TS touch probe in any probing direction.

Touch probe cycle 4 measures any position on the workpiece in the probing direction defined by a vector. Unlike other measuring cycles, Cycle 4 enables you to enter the measuring distance and feed rate directly. Also, the touch probe retracts by a definable value after determining the measured value.

1. The TNC moves from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).

2. After the TNC has saved the position, the TNC stops the probing motion. The TNC saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.

3. Finally, the TNC performs a positioning movement in the direction opposite to the direction of probing. You define the traverse path in parameter MB—the touch probe is moved to a point no farther than the starting point.

Please note while programming:

The TNC retracts the touch probe by no more than the retraction distance MB and does not pass the starting point of the measurement. This rules out any collision during retraction.

Ensure during pre-positioning that the TNC moves the probe-tip center without compensation to the defined position!

Remember that the TNC always writes to four successive Q parameters. If the TNC was not able to determine a valid touch point, the 4th result parameter will have the value –1.
17 Touch Probe Cycles: Special Functions
17.3 MEASURING IN 3-D (Cycle 4)

Cycle parameters

- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters. Input range: 0 to 1999

- **Relative measuring path in X**: X component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999

- **Relative measuring path in Y**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999

- **Relative measuring path in Z**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999

- **Maximum measuring path**: Enter the maximum distance from the starting point by which the touch probe may move along the direction vector. Input range -99999.9999 to 99999.9999

- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000

- **Maximum retraction distance**: Traverse path in the direction opposite the probing direction, after the stylus was deflected. Input range 0 to 99999.9999

- **Reference system? (0=ACTUAL/1=REF)**: Define whether the probing result should be saved in the input coordinate system (ACTUAL) or referenced to the machine-table coordinate system (REF):
  - 0: Save the measured result in the ACTUAL system
  - 1: Save the measured result in the REF system

NC blocks

<table>
<thead>
<tr>
<th>Block</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 TCH PROBE 4.0</td>
<td>MEASURING IN 3-D</td>
</tr>
<tr>
<td>5 TCH PROBE 4.1</td>
<td>Q1</td>
</tr>
<tr>
<td>6 TCH PROBE 4.2</td>
<td>IX-0.5 IY-1 IZ-1</td>
</tr>
<tr>
<td>7 TCH PROBE 4.3</td>
<td>DIST+45 F100 MB50</td>
</tr>
<tr>
<td>REFERENCE SYSTEM:0</td>
<td></td>
</tr>
</tbody>
</table>
Calibrating a touch trigger probe

17.4  Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe, otherwise the TNC cannot provide precise measuring results.

- Always calibrate a touch probe in the following cases:
  - Commissioning
  - Stylus breakage
  - Stylus exchange
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Change of active tool axis

The TNC assumes the calibration values for the active probe system directly after the calibration process. The updated tool data become effective immediately, and a new tool call is not necessary.

During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The TNC provides calibration cycles for calibrating the length and the radius:

- Press the TOUCH PROBE soft key
  - Display the calibration cycles: Press CALIBRATE TS
  - Select the calibration cycle

### Calibration cycles of the TNC

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Function</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Calibrating the length</td>
<td>589</td>
</tr>
<tr>
<td></td>
<td>Measure the radius and the center offset using a calibration ring</td>
<td>591</td>
</tr>
<tr>
<td></td>
<td>Measure the radius and the center offset using a stud or a calibration pin</td>
<td>593</td>
</tr>
<tr>
<td></td>
<td>Measure the radius and the center offset using a calibration sphere</td>
<td>587</td>
</tr>
</tbody>
</table>
17.5 Displaying calibration values

The TNC saves the effective length and effective radius of the touch probe in the tool table. The TNC saves the ball-tip center offset of the touch probe in the touch-probe table in the CAL_OF1 (principal axis) and CAL_OF2 (secondary axis) columns. You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs. When running a touch probe cycle in the Manual Operation mode, the TNC saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC: *.

Please make sure the correct tool number is active when you use the touch probe system. Regardless of whether you want to use a touch probe cycle in automatic mode or Manual operation mode.

For more information, see Chapter Touch probe table.
CALIBRATE TS (Cycle 460, DIN/ISO: G460)

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically on an exact calibration sphere. You can do radius calibration alone, or radius and length calibration.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

1. Clamp the calibration sphere and check for potential collisions.
2. In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3. The first movement in the cycle is in the negative direction of the touch probe axis.
4. Then the cycle determines the exact center of the sphere in the touch probe axis.

Please note while programming:

HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Pre-position the touch probe in the program so that it is located approximately above the center of the calibration sphere.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.
**Touch Probe Cycles: Special Functions**

**17.6 CALIBRATE TS (Cycle 460, DIN/ISO: G460)**

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP in the touch probe table. Input range 0 to 99999.9999
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points
- **Number of probe points in the plane (4/3)** Q423: Number of measuring points on the diameter. Input range 0 to 8
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Calibrate length** (0/1) Q433: Define whether the TNC should also calibrate the touch probe length after radius calibration:
  - 0: Do not calibrate touch probe length
  - 1: Calibrate touch probe length
- **Datum for length** Q434 (absolute): Coordinate of the calibration sphere center. The definition is only required if length calibration is to be carried out. Input range -99999.9999 to 99999.9999

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>S T C H P R O B E 4 6 0 C A L I B R A T E T S</td>
</tr>
<tr>
<td>Q407=12.5 ;SPHERE RADIUS</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q301=1 ;MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q423=4 ;NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q380=+0 ;REFERENCE ANGLE</td>
</tr>
<tr>
<td>Q433=0 ;CALIBRATE LENGTH</td>
</tr>
<tr>
<td>Q434=-2.5 ;DATUM</td>
</tr>
</tbody>
</table>
17.7 CALIBRATE TS LENGTH (Cycle 461, DIN/ISO: G461)

Cycle run
Before starting the calibration cycle, you must set the datum in the spindle axis so that Z=0 on the machine table; you must also preposition the touch probe over the calibration ring.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

1. The TNC orients the touch probe to the angle CAL_ANG from the touch probe table (only if your touch probe can be oriented).
2. The TNC probes from the current position in a negative spindle axis direction at the probing feed rate (column F from the touch probe table).
3. The TNC then returns the touch probe at rapid traverse (column FMAX from the touch probe table) to the start position.
Please note while programming:

HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

 Datum Q434 (absolute): Datum for the length (e.g. height of the ring gauge). Input range -99999.9999 to 99999.9999

NC blocks

```
5 TCH PROBE 461 CALIBRATE TS LENGTH
Q434=+5 ;DATUM
```
17.8 CALIBRATE TS RADIUS INSIDE
(Cycle 462, DIN/ISO: G462)

**Cycle run**
Before starting the calibration cycle, you need to preposition the touch probe in the center of the calibration ring and at the required measuring height.

When calibrating the ball tip radius, the TNC executes an automatic probing routine. During the first probing cycle, the TNC determines the center of the calibration ring or stud (coarse measurement) and positions the touch probe in the center. Then the ball tip radius is determined during the actual calibration process (fine measurement). If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The touch probe orientation determines the calibration routine:
- **No orientation possible or orientation possible in only one direction:** The TNC executes one approximate and one fine measurement and determines the effective ball tip radius (column R in tool.t).
- **Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable):** The TNC executes one approximate and one fine measurement, rotates the touch probe by 180° and then executes four more probing operations. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- **Any orientation possible (e.g. HEIDENHAIN infrared touch probes):** For probing routine, see "orientation possible in two directions."
Please note while programming:

HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The center offset can be determined only with a suitable touch probe. A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information. The characteristic of whether and how your touch probe can be oriented is already defined in HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

- **RING RADIUS** Q407: Diameter of the ring gauge. Input range 0 to 99.9999
- **SET-UP CLEARANCE** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999
- **NO. OF PROBE POINTS** Q407 (absolute): Number of measuring points on the diameter. Input range 0 to 8
- **REFERENCE ANGLE** Q380 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range 0 to 360.0000

**NC blocks**

<table>
<thead>
<tr>
<th>5 TCH PROBE 462 TS CALIBRATE IN RING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q407=+5 ;RING RADIUS</td>
</tr>
<tr>
<td>Q320=+0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q423=+8 ;NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q380=+0 ;REFERENCE ANGLE</td>
</tr>
</tbody>
</table>
17.9  CALIBRATE TS RADIUS OUTSIDE
(Cycle 463, DIN/ISO: G463)

Cycle run
Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration pin. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration pin.

When calibrating the ball tip radius, the TNC executes an automatic probing routine. During the first probing cycle, the TNC determines the center of the calibration ring or stud (coarse measurement) and positions the touch probe in the center. Then the ball tip radius is determined during the actual calibration process (fine measurement). If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If a program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The touch probe orientation determines the calibration routine:
- No orientation possible or orientation possible in only one direction: The TNC executes one approximate and one fine measurement and determines the effective ball tip radius (column R in tool.t)
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The TNC executes one approximate and one fine measurement, rotates the touch probe by 180° and then executes four more probing operations. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): For probing routine, see "orientation possible in two directions."
Touch Probe Cycles: Special Functions
17.9 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, DIN/ISO: G463)

Please note while programming:

- HEIDENHAIN only warrants the function of the touch probe cycles if HEIDENHAIN touch probes are used.

- Before a cycle definition you must have programmed a tool call to define the touch probe axis. The center offset can be determined only with a suitable touch probe. A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

- In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information. The characteristic of whether and how your touch probe can be oriented is already defined in HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

- **STUD RADIUS** Q407: Diameter of the ring gauge. Input range 0 to 99.9999

- **SET-UP CLEARANCE** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table). Input range 0 to 99999.9999

- **MOVE TO CLEARANCE** Q301: Definition of how the touch probe is to move between the measuring points:
  - 0: Move at measuring height between measuring points
  - 1: Move at clearance height between measuring points

- **NO. OF PROBE POINTS** Q407 (absolute): Number of measuring points on the diameter. Input range 0 to 8

- **REFERENCE ANGLE** Q380 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range 0 to 360.0000

### NC blocks

<table>
<thead>
<tr>
<th>Q407</th>
<th>STUD RADIUS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q320</td>
<td>SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q301</td>
<td>MOVE TO CLEARANCE</td>
</tr>
<tr>
<td>Q423</td>
<td>NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q380</td>
<td>REFERENCE ANGLE</td>
</tr>
</tbody>
</table>
Visual Setup
Control VSC
(software option 136)
18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Fundamentals
In order to use the Visual Setup Control you require the following components:
- Software: Option 136 Visual Setup Control (VSC)
- Hardware: Camera system of HEIDENHAIN

Application
Camera-based checking of the setup situation (option number 136 Visual Setup Control) allows you to monitor the current setup situation before and during processing, and to compare this with a safe target status. After setup, simple cycles for automatic monitoring are available.

Reference images of the current working space are recorded in the camera system. With cycles 600 GLOBAL WORKING SPACE or 601 LOCAL WORKING SPACE, the TNC produces an image of the working space and compares the image with previously prepared reference images. These cycles can highlight irregularities in the working space. The operator decides if the NC program should be suspended or continued in the event of a fault.

Using VSC offers the following advantages:
- The control can recognize elements (e.g. tools or chucking equipment) located in the working space after the start of the program
- If you would always like to clamp a workpiece in the same position (e.g. hole top right), the control can check the clamping situation
- For documentation purposes you can produce an image of the current working space (e.g. of a rarely-needed clamping situation)

Terms
The following terms are used in connection with VSC:

<table>
<thead>
<tr>
<th>Term</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference image</td>
<td>Saved image, highlighted as a reference. A reference image shows a situation in the working space that you regard as safe. Therefore only create reference images showing safe, non-hazardous situations.</td>
</tr>
<tr>
<td>Mid-value image</td>
<td>The control creates a mid-value image taking into account all reference images. The control compares new images with the mid-value image as part of evaluation.</td>
</tr>
</tbody>
</table>
## Camera-based monitoring of the setup situation VSC (option number 136)

<table>
<thead>
<tr>
<th>Term</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Error</td>
<td>If you take a picture that shows a poor situation (e.g., workpiece wrongly clamped), you can create what is known as an error image. It is not advisable to highlight an error image as a reference image.</td>
</tr>
<tr>
<td>Monitoring area</td>
<td>Denotes an area that you highlight with the mouse. When evaluating new images, the control only refers to this area. Parts of images outside the monitoring area have no effect on the results of the monitoring process. Several monitoring areas can be defined. Monitoring areas are not linked to images.</td>
</tr>
<tr>
<td>Error</td>
<td>Area on an image containing a deviation from the desired position. Errors always relate to the image to which they were saved (error image) or to the last image evaluated.</td>
</tr>
<tr>
<td>Monitoring phase</td>
<td>No further reference images are produced in the monitoring phase. You can use the cycle for automatic monitoring of your working space. In this phase, the control only issues a warning if it finds a deviation when comparing images.</td>
</tr>
</tbody>
</table>
Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Produce live image

In the **Manual operation** mode, you can display and save the current camera view as a live image. The control only uses the picture taken here for automatic monitoring of the setup situation. Images produced in this menu may be used for documentation and traceability. For example, you could record the current setup situation. The control saves the image produced as a .png file in `TNC:\system\visontool\live_view`. The name of the saved image is made up of the date and time it was created.

Procedure

Proceed as follows to save the camera’s live image:

- Press the **CAMERA** soft key
- Press the **LIVE IMAGE** soft key: The TNC shows you the current camera view
- Press the **SAVE IMAGE** soft key: Create a live image from the current camera view

Options in Live Image mode

The control provides the following options:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>LIGHTER</strong></td>
<td>Increase camera brightness</td>
</tr>
<tr>
<td>The settings made here only affect Live Image mode, and have no influence on pictures taken in automatic mode.</td>
<td></td>
</tr>
<tr>
<td><strong>DARKER</strong></td>
<td>Reduce camera brightness</td>
</tr>
<tr>
<td>The settings made here only affect Live Image mode, and have no influence on pictures taken in automatic mode.</td>
<td></td>
</tr>
<tr>
<td><strong>GO BACK</strong></td>
<td>Go back to the previous screen</td>
</tr>
</tbody>
</table>
Manage monitoring data

In the Manual operation mode you can manage images from cycles 600 and 601.

Proceed as follows to enter the monitoring data:

- Press the CAMERA soft key
- Press the MONITORING DATA MANAGEMENT soft key: The control shows a list of the NC programs monitored
- Press the OPEN soft key: The control shows a list of monitoring points
- Edit the desired data

Select data

You can select the marked interfaces 1 with the mouse. These interfaces make it easier to search and show results in a manageable way.

- **All images**: Display all images for this monitoring file
- **Reference images**: Only display reference images
- **Images with error**: Display all images where you have highlighted an error
Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Features of the monitoring data management

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>REFERENCE I</td>
<td>Mark selected image as a reference image</td>
</tr>
<tr>
<td>IMAGE</td>
<td>Please note: A reference image shows a situation in the working space that you regard as safe.</td>
</tr>
<tr>
<td></td>
<td>All reference images are used as part of the evaluation process. If you add or remove an image as a reference image, this has an effect on the results of image evaluation.</td>
</tr>
<tr>
<td>DELETE I</td>
<td>Delete image currently selected</td>
</tr>
<tr>
<td>EVALUATE ALL</td>
<td>Carry out automatic image evaluation</td>
</tr>
<tr>
<td>IMAGERES</td>
<td>The control carries out an image evaluation according to the reference images and the monitoring areas.</td>
</tr>
<tr>
<td>CONFIGURE</td>
<td>Change monitoring area or highlight an error</td>
</tr>
<tr>
<td>GO BACK</td>
<td>Go back to the previous screen</td>
</tr>
</tbody>
</table>

Further Information: Configuration, page

If you change the configuration, the control carries out an image evaluation.
Overview

The TNC provides two cycles you can use for visual setup control in the **Programming** mode of operation:

- The soft-key row shows all available touch probe functions divided into groups.
- Chose soft key MONITORING WITH CAMERA

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>600</td>
<td>600 GLOBAL WORKING SPACE</td>
<td>606</td>
</tr>
<tr>
<td>601</td>
<td>601 LOCAL WORKING SPACE</td>
<td>611</td>
</tr>
</tbody>
</table>
18 Visual Setup Control VSC (software option 136)

18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Results of the image evaluation

The results of the image evaluation depend on the monitoring area and the reference images. When evaluating all images, each image is evaluated according to the current configuration and the results are compared with the data last saved.

If you change the monitoring area or add/delete reference images, the images may be tagged with the following symbol:

- **Triangle**: You have changed the monitoring data, e.g. tagged an image with errors as a reference image or deleted a monitoring area. This makes the monitoring less sensitive.
  
  This has an effect on your reference images and on the mid-value image. The changes to the configuration mean that the control can no longer recognize errors which were previously saved to this image! If you would like to continue, confirm the reduced monitoring sensitivity and the new settings will be accepted.

- **Solid circle**: You have changed the monitoring data, monitoring is more sensitive.

- **Blank circle**: No error message: All deviations saved in the image have been recognized, monitoring has not identified any conflicts.

Configuration

You are now able to change your settings regarding the monitoring area and error area at any time. By pressing the soft key **CONFIGURE** the soft key row is toggled and you can change the settings.

- You also are now able to edit your previously entered settings. If you carry out the change in this menu, the result of the image evaluation may change. The same monitoring area applies for all reference images. (More information, see "Results of the image evaluation", page 602.)
Camera-based monitoring of the setup situation VSC (option number 136)

- You are able to click on the image and draw up a rectangular frame. This way you define the monitoring area. (More information, see “Fundamentals”, page 596.) If you define monitoring areas in a setting that is always exposed or in which differences in contrast are to be expected, false alerts will be displayed. If you draw a new monitoring area or change or delete an already drawn up monitoring area, this will affect the result of the image evaluation. Due to the changed settings the TNC must check whether these changes may have an impact on the previously generated images.

- You are able to click on the image and draw up a rectangular frame. This way you define a new area with errors. The area is marked red. It is recommended to mark only errors that may reoccur exactly at this spot. It is not advisable to mark areas that are contaminated with chips or drilling fluid. The errors need to be reproducible in the exact same way. (More information, see “Fundamentals”, page 596.) If you define monitoring areas in a setting that is always exposed or in which differences in contrast are to be expected, false alerts will be displayed. If you draw a new area with errors or change or delete an already drawn up area of errors, this will affect the result of the image evaluation. Due to the changed settings the TNC must check whether these changes may have an impact on the previously generated images. You can also draw several areas with errors. It is not advisable to draw errors onto reference images.

- The TNC checks whether and how the new settings affect the new image: (More information, see “Results of the image evaluation”, page 602)

- The TNC checks whether and how the new settings affect the new image: (More information, see “Results of the image evaluation”, page 602)

- Save current image and return to previous screen. If you changed the configuration, the TNC will perform an image evaluation. (More information, see “Results of the image evaluation”, page 602.)

- You discard all changes and return to the previously displayed screen.
18.1 Camera-based monitoring of the setup situation VSC (option number 136)

Defining the monitoring area

The monitoring area is defined with the modes of operation Single Block or Block Scan. The TNC will prompt you to define a monitoring area. The TNC will display this prompt on the screen after you have started the cycle for the first time in the modes of operation Single Block or Block Scan.

A monitoring area consists of one or more windows that you draw with your mouse. The TNC will only scan these areas of the image. Errors outside of the monitoring area will not be detected. The monitoring area is not linked to images but only with the appropriate monitoring file QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Monitoring areas may overlap.

Definition of the monitoring area:
1. Click with the mouse on the image and draw a new area
2. If you would like to define more windows press the soft key DRAW AREA and repeat the operation at the appropriate position

After you have defined the monitoring area you press the following soft key, for example:

- Save current image and return to previous screen.

The message is displayed: Monitoring point configured: Choose soft key!

The status display at the top right of the image will display information about the minimum number of reference images, about the current number of reference images and the current number of error images.
Possible queries

The cycles of the workspace monitoring enter a value in parameter Q601.
The following values are possible:
- Q601 = 1: No error
- Q601 = 2: Error
- Q601 = 3: You have yet not defined a monitoring area or you did not save enough reference images
- Q601 = 10: Internal error (no signal, faulty camera, etc.)

You can use parameter Q601 for internal queries.

More information about If-Then Decisions with Q parameter functions are found in the user manuals of the ‘TNC 640, section 9.6’

Here, a possible programming example for a query:

```
0 BEGIN PGM 5MM
1 BLK FORM CYLINDER Z R42 L150  Blank form Cylinder
2 FUNCTION MODE MILL            Activate milling mode
3 TCH PROBE 601 WORKSPACE LOCAL Define Cycle 600
   QS600 = O5 ;MONITORING POINT
   Q309 = +0 ;PGM STOP IF ERROR
   Q613 = +0 ;KEEP CAMERA LID OPEN
   Q617 = 10 ;REFERENCE IMAGES
4 FN 9: IF Q601 EQU 1 GOTO LBL 20  If parameter Q601 = 1, then jump to LBL 20
5 FN 9: IF Q601 EQU 2 GOTO LBL 21  If parameter Q601 = 2, then jump to LBL 21
6 FN 9: IF Q601 EQU 3 GOTO LBL 22  If parameter Q601 = 3, then jump to LBL 22
7 FN 9: IF Q601 EQU 10 GOTO LBL 23 If parameter Q601 = 10, then jump to LBL 23
8 TOOL CALL “GEAR_HOB_D75”       Call the tool
9 L X+... Y+... R0 FMAX          Program the machining
   ...
   ...
57 LBL 21                        Definition LBL 21
58 STOP                          Program stop, the operator can check the condition of the workspace
59 LBL 0
60 END PGM 5MM
```
18.2 Workspace Global (Cycle 600)

Application

With Cycle 600, Workspace Global, you monitor the workspace of your tooling machine. The TNC will generate an image of the current workspace from a position determined by your machine tool builder. Then, the TNC will match the image with previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 600 is valid once specified and must not be called up. Before you work with the visual setup control you must create reference images (more information, see "Creating reference images", page 606) and define a monitoring area (more information, see "Monitoring phase", page 609).

Creating reference images

The TNC will start generating reference images as soon as you run the cycle for the first time in the program modes Single Block or Block Scan.

The following cycle process applies as long as the TNC still requires reference images to be saved. Specify the number of reference images with parameter Q617.

Cycle run

1. The camera will be mounted by the machine tool builder onto the main spindle.
2. The TNC automatically opens the camera lid.
3. The TNC will generate an image of the current condition and display it on the screen.
4. During the first run of this cycle a message is displayed at the bottom of the screen "Monitoring point not configured: Draw areas!"
5. Define the monitoring area. (More information, see "Defining the monitoring area", page 604)
6. You can decide whether the current image should be saved as a reference image or an error image, but you can also change the monitoring area. (More information, see "Configuration", page 602).
7. Press the soft key. BACK.
8. To complete the operation the TNC will close the camera lid.
9. Press the NC start and run your program as usual.

After you have defined the monitoring area, you can press in addition of the soft key: BACK you can also choose the following soft keys:
The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (More information, "Results of the image evaluation")

The status display at the top right displays the word “Reference”. You have marked the current image as the reference image. Because a reference image can never be an error image at the same time, the soft key ERROR IMAGE is gray. (More information, see "Fundamentals", page 596)

The status display at the top right displays the word “Error”. You have marked the current image as the error image. Because an error image can never be a reference image at the same time, the soft key ERROR IMAGE is gray. (More information, see "Fundamentals", page 596)

The soft key row changes. You are now able to change your previously entered settings with respect to the monitoring area and the sensitivity. If you carry out the change in this menu, this may impact all of your images. (More information, see "Configuration", page 602)

The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (More information, "Results of the image evaluation")

As soon as the TNC generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected the following message is displayed: Too few reference images: Choose next action with soft key! This message will no longer be displayed if the number of reference images defined in parameter Q617 has been reached.

The TNC will generate mean image taking into account all reference images. New images are compared during evaluation with the mean image taking into account variance. Only after all reference images are available the cycle will no longer stop because of too few reference images.
Defining the monitoring area
The monitoring area is defined with the modes of operation Single Block or Block Scan. The TNC will prompt you to define a monitoring area. The TNC will display this prompt on the screen after you have started the cycle for the first time in the modes of operation Single Block or Block Scan.
A monitoring area consists of one or more windows that you draw with your mouse. The TNC will only scan these areas of the image. Errors outside of the monitoring area will not be detected. The monitoring area is not linked to images but only with the appropriate monitoring file QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.
Monitoring areas may overlap.

Definition of the monitoring area:
1. Click with the mouse on the image and draw a new area
2. If you would like to define more windows press the soft key DRAW AREA and repeat the operation at the appropriate position

After you have defined the monitoring area you press the following soft key, for example:

- Save current image and return to previous screen.

The message is displayed: Monitoring point configured: Choose soft key!

The status display at the top right of the image will display information about the minimum number of reference images, about the current number of reference images and the current number of error images.
Monitoring phase

Cycle run: Monitoring phase
1. The camera will be mounted by the machine tool builder onto the main spindle. The main spindle moves to a position defined by the machine tool builder.
2. After the TNC has reached this position, it will automatically open the camera lid.
3. The TNC will generate an image of the current condition.
4. Then it will match the image against the mean value and the variance image (more information, see “Fundamentals”, page 596).
5. Depending on whether the TNC detected a so-called “Error” (deviation), it may now enforce a program stop (more information, see “Fundamentals”, page 596). If parameter Q309=1 is set, the TNC will display the image on the screen upon detecting an error. If parameter Q309=0 is set, no image will be displayed on the screen and the program will not be stopped.
6. To complete the operation the TNC will close the camera lid.

Please note while programming:

- In addition to the attribute Reference image, you can also assign the attribute Error image to your images. This assignment may impact the image evaluation.
  - Remember the following:
    - A reference image may never be an error image at the same time.

- If you change the monitoring area, this has an impact on all images.
  - Ideally, you define the monitoring area only once at the beginning and then make no or only little changes to it.

- The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.
  - Specify a sensible number of reference images in parameter Q617. (target value: 10 images).
  - You can also create more reference images than you have specified in Q617.

- Your machine must be prepared for the workspace monitoring!
Danger of contaminating the camera due to open camera lid with parameter Q613. This could lead to blurred images, the camera may be damaged. Close the camera lid before continuing with the process.

Danger of collision during automatic positioning of the camera. The camera and your machine may be damaged. Consult your machine tool builder, at which point the TNC will preposition the camera. Your machine tool builder specifies at which coordinates Cycle 600 is positioned.

**Cycle parameters**

- **Monitoring point** Q600 (string parameter): Enter the name of your monitoring file
- **Position feed rate** Q616: Feed rate the TNC uses to position the camera. Here, TNC moves to a position defined by the machine tool builder.
- **PGM stop if error** Q309: (0/1) Definition of whether the TNC will stop the program after detecting an error.
  - 0: Program will not stop after detecting an error. Even if not all reference images have been generated, the program will not stop. As a result, the generated image will not be displayed in the screen. Parameter Q601 is also described in Q309=0.
  - 1: Program will stop after detecting an error, the generated image will be displayed in the screen. If not enough reference images have been generated yet, each new image will be displayed on the screen until the TNC has generated enough reference images. If an error is detected, the TNC will display a message.
- **Reference images** Q617: Number of reference images that are required by the TNC for monitoring.

**NC blocks**

<table>
<thead>
<tr>
<th>4 TCH PROBE 600 WORKSPACE GLOBAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q600=&quot;OS&quot;;MONITORING POINT</td>
</tr>
<tr>
<td>Q616=500 ;POSITION FEED RATE</td>
</tr>
<tr>
<td>Q309=1 ;PGM STOP IF ERROR</td>
</tr>
<tr>
<td>Q617=10 ;REFERENCE IMAGES</td>
</tr>
</tbody>
</table>
18.3 Workspace Local (Cycle 601)

Application
With Cycle 601, Workspace Local, you monitor the workspace of your tooling machine. The TNC will generate an image of the current workspace from the position of the spindle at the point in time of the cycle call. Then, the TNC will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 601 is valid once specified and must not be called up. Before you work with the visual setup control you must create reference images (more information, see "Creating reference images", page 611) and define a monitoring area (more information, see "Monitoring phase", page 614).

Creating reference images
The TNC will start generating reference images as soon as you run the cycle for the first time in the program modes Single Block or Block Scan.
The following cycle process applies as long as the TNC still requires reference images to be saved. Specify the number of reference images with parameter Q617.

Cycle run
1 The camera will be mounted by the machine tool builder onto the main spindle.
2 The TNC automatically opens the camera lid.
3 The TNC will generate an image of the current condition and display it on the screen.
4 During the first run of this cycle a message is displayed at the bottom of the screen "Monitoring point not configured: Draw areas!"
5 Define the monitoring area. (More information, see "Defining the monitoring area", page 604)
6 You can decide whether the current image should be saved as a reference image or an error image, but you can also change the monitoring area. (More information, see "Configuration", page 602).
7 Press the soft key. BACK.
8 To complete the operation the TNC will close the camera lid.
9 Press the NC start and run your program as usual.
After you have defined the monitoring area, you can press in addition of the soft key: BACK you can also choose the following soft keys:
The TNC will save the current image and return to the program run screen. If you changed the configuration, the TNC will perform an image evaluation. (More information, “Results of the image evaluation”)

The status display at the top right displays the word “Reference”. You have marked the current image as the reference image. Because a reference image can never be an error image at the same time, the soft key ERROR IMAGE is gray. (More information, see “Fundamentals”, page 596)

The status display at the top right displays the word “Error”. You have marked the current image as the error image. Because an error image can never be a reference image at the same time, the soft key ERROR IMAGE is gray. (More information, see “Fundamentals”, page 596)

The soft key row changes. You are now able to change your previously entered settings with respect to the monitoring area and the sensitivity. If you carry out the change in this menu, this may impact all of your images. (More information, see “Configuration”, page 602)

As soon as the TNC generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected the following message is displayed: **Too few reference images: Choose next action with soft key!**. This message will no longer be displayed if the number of reference images defined in parameter Q617 has been reached.

The TNC will generate mean image taking into account all reference images. New images are compared during evaluation with the mean image taking into account variance. Only after all reference images are available the cycle will no longer stop because of too few reference images.
Defining the monitoring area

The monitoring area is defined with the modes of operation Single Block or Block Scan. The TNC will prompt you to define a monitoring area. The TNC will display this prompt on the screen after you have started the cycle for the first time in the modes of operation Single Block or Block Scan.

A monitoring area consists of one or more windows that you draw with your mouse. The TNC will only scan these areas of the image. Errors outside of the monitoring area will not be detected. The monitoring area is not linked to images but only with the appropriate monitoring file QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Monitoring areas may overlap.

Definition of the monitoring area:

1. Click with the mouse on the image and draw a new area
2. If you would like to define more windows press the soft key DRAW AREA and repeat the operation at the appropriate position

After you have defined the monitoring area you press the following soft key, for example:

- Save current image and return to previous screen.

The message is displayed: Monitoring point configured: Choose soft key!

The status display at the top right of the image will display information about the minimum number of reference images, about the current number of reference images and the current number of error images.
Monitoring phase
The monitoring phase starts as soon as the TNC has generated enough reference images.

Cycle run: Monitoring phase
1. The camera will be mounted by the machine tool builder onto the main spindle.
2. The TNC automatically opens the camera lid.
3. The TNC will generate an image of the current condition.
4. Then it will match the image against the mean value and the variance image (more information, see "Fundamentals", page 596)
5. Depending on whether the TNC detected a so-called "error" (deviation), it may now enforce a program stop (more information, "Results of the image evaluation") If parameter Q309=1 is set, the TNC will display the image on the screen upon detecting an error. If parameter Q309=0 is set, no image will be displayed on the screen and the program will not be stopped.
6. Depending on parameter Q613, the TNC either leaves the camera lid open or closes it.

Please note while programming:

- In addition to the attribute Reference image, you can also assign the attribute Error image to your images. This assignment may impact the image evaluation.
  Remember the following:
  ▶ A reference image may never be an error image at the same time.

- If you change the monitoring area, this has an impact on all images.
  ▶ Ideally, you define the monitoring area only once at the beginning and then make no or only little changes to it.

- The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.
  ▶ Specify a sensible number of reference images in parameter Q617. (target value: 10 images)
  ▶ You can also create more reference images than you have specified in Q617.

- Your machine must be prepared for the workspace monitoring!
Danger of contaminating the camera due to open camera lid with parameter Q613. This could lead to blurred images, the camera may be damaged. Close the camera lid before continuing with the process!

**Cycle parameters**

- **Monitoring point** Q600 (string parameter): Enter the name of your monitoring file.

- **PGM stop if error** Q309: (0/1) Definition of whether the TNC will stop the program after detecting an error.
  - 0: Program will not stop after detecting an error. Even if not all reference images have been generated, the program will not stop. As a result, the generated image will not be displayed in the screen. Parameter Q601 is also described in Q309=0.
  - 1: Program will stop after detecting an error, the generated image will be displayed in the screen. If not enough reference images have been generated yet, each new image will be displayed on the screen until the TNC has generated enough reference images. If an error is detected, the TNC will display a message.

- **Keep camera open** Q613: (0/1) Definition of whether the TNC should close the camera lid after monitoring.
  - 0: The TNC closes the camera lid after running Cycle 601.
  - 1: The TNC keeps camera lid open after running Cycle 601. This function is only recommended if you would like to generate yet another image of the workspace in a different position after the first cycle call of Cycle 601. In order to so, program the new position on a straight line block and then call up Cycle 601 with a new monitoring point. Program Q613=0 before continuing with the process!

- **Reference images** Q617: Number of reference images that are required by the TNC for monitoring.

**NC blocks**

<table>
<thead>
<tr>
<th>4 TCH PROBE 601 WORKSPACE LOCAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q600=&quot;OS&quot;;MONITORING POINT</td>
</tr>
<tr>
<td>Q309=+1 ;PGM STOP IF ERROR</td>
</tr>
<tr>
<td>Q613=0 ;KEEP CAMERA OPEN</td>
</tr>
<tr>
<td>Q617=10 ;REFERENCE IMAGES</td>
</tr>
</tbody>
</table>
19

Touch Probe Cycles: Automatic Kinematics Measurement
Touch Probe Cycles: Automatic Kinematics Measurement

19.1 Kinematics Measurement with TS Touch Probes (KinematicsOpt option)

Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts need to be manufactured with precision and reproducible accuracy even over long periods.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see 1 in the figure at right), and the kinematic conditions actually existing on the machine (see 2 in the figure at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see 3 in the figure at right). It is therefore necessary for the model to approach reality as closely as possible.

The TNC function KinematicsOpt is an important component that helps you to really fulfill these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are in the form of tables or spindle heads. A calibration sphere is fixed at any position on the machine table, and measured with a resolution that you define. During cycle definition you simply define for each rotary axis the area that you want to measure.

From the measured values, the TNC calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematic table.
### Overview

The TNC offers cycles that enable you to automatically save, check and optimize the machine kinematics:

<table>
<thead>
<tr>
<th>Soft key</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>450</td>
<td>SAVE KINEMATICS</td>
<td>621</td>
</tr>
<tr>
<td></td>
<td>Automatically saving and restoring kinematic configurations</td>
<td></td>
</tr>
<tr>
<td>451</td>
<td>MEASURE KINEMATICS</td>
<td>624</td>
</tr>
<tr>
<td></td>
<td>Automatically checking or optimizing the machine kinematics</td>
<td></td>
</tr>
<tr>
<td>452</td>
<td>PRESET COMPENSATION</td>
<td>638</td>
</tr>
<tr>
<td></td>
<td>Automatically checking or optimizing the machine kinematics</td>
<td></td>
</tr>
</tbody>
</table>
19.2 Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt), 8 (Software option 1) and 17 (Touch Probe function) must be enabled.
- The 3-D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table. HEIDENHAIN recommends using the calibration spheres KKH 250 (ID number 655475-01) or KKH 100 (ID number 655475-02), which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.
- The kinematics description of the machine must be complete and correct. The transformation values must be entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- The machine tool builder must have saved the machine parameters for CfgKinematicsOpt in the configuration data. maxModification specifies the tolerance limit from which the TNC should indicate if the modifications to kinematic data are above this limit value. maxDevCalBall specifies how large the measured calibration sphere radius should be from the cycle parameters entered. mStrobeRotAxPos defines an M function specified by the machine tool builder for positioning the rotary axes.

Please note while programming:

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

If an M function has been defined in machine parameter mStrobeRotAxPos, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450). If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.
19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option)

Cycle run
With the touch probe cycle 450 you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.

Please note while programming:

- Always save the active kinematics configuration before running a kinematics optimization. Advantage:
  - You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).
- With the Restore mode, note that
  - the TNC can restore saved data only to a matching kinematic configuration.
  - a change in the kinematics always changes the preset as well. Set the preset again if necessary.
### Touch Probe Cycles: Automatic Kinematics Measurement

#### 19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option)

**Cycle parameters**

- **Mode (0/1/2/3)**
  - Q410: Define if you wish to backup or restore the kinematics:
    - 0: Backup active kinematics
    - 1: Restore saved kinematics
    - 2: Display current memory status
    - 3: Delete a data record
  
- **Memory designation**
  - Q409/QS409: Number or name of the data block designator. For a number, enter a value from 0 to 99999; for a name, enter a maximum of 16 characters. 16 memory spaces in total are available. Q409 has no function if Mode 2 has been selected. Wildcards can be used for searching in Modes 1 and 3 (Restore and Delete). If the TNC finds several possible data blocks because of the wildcards, it restores the mean values of the data (Mode 1) or deletes all selected data blocks after confirmation (Mode 3). You can use the following wildcards in a search:
    - ?: A single indefinite character
    - $: A single alphabetic character (letter)
    - #: A single indefinite number
    - *: An indefinite character string of any length

**Saving the current kinematics**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 450 SAVE KINEMATICS</td>
<td>Q410=0 ;MODE</td>
</tr>
<tr>
<td>Q409=947 ;MEMORY DESIGNATION</td>
<td></td>
</tr>
</tbody>
</table>

**Restoring data blocks**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 450 SAVE KINEMATICS</td>
<td>Q410=1 ;MODE</td>
</tr>
<tr>
<td>Q409=948 ;MEMORY DESIGNATION</td>
<td></td>
</tr>
</tbody>
</table>

**Displaying all saved data blocks**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 450 SAVE KINEMATICS</td>
<td>Q410=2 ;MODE</td>
</tr>
<tr>
<td>Q409=949 ;MEMORY DESIGNATION</td>
<td></td>
</tr>
</tbody>
</table>

**Deleting data blocks**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 TCH PROBE 450 SAVE KINEMATICS</td>
<td>Q410=3 ;MODE</td>
</tr>
<tr>
<td>Q409=950 ;MEMORY DESIGNATION</td>
<td></td>
</tr>
</tbody>
</table>

**Logging function**

After running Cycle 450, the TNC will create a measuring log (TCHPRAUTO.HTML) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

The other data in the log vary depending on the selected mode:

- **Mode 0**: Logging of all axis entries and transformation entries of the kinematics chain that the TNC has saved.
- **Mode 1**: Logging of all transformation entries before and after restoring the kinematics configuration.
- **Mode 2**: List of the saved data records.
- **Mode 3**: List of the deleted data records.
Notes on data management

The TNC stores the saved data in the file `TNC:\table\DATA450.KD`. This file can be backed up on an external PC with `TNCREMO`, for example. If the file is deleted, the stored data are removed, too. If the data in the file are changed manually, the data records can become corrupted so that they cannot be used anymore.

If the `TNC:\table\DATA450.KD` file does not exist, it is generated automatically when Cycle 450 is executed.

Make sure that you delete empty files with the name `TNC:\table\DATA450.KD`, if any, before starting Cycle 450. If there is an empty storage table (`TNC:\table\DATA450.KD`) that does not contain any lines yet, an error message will be displayed upon starting to run Cycle 450. In this case, delete the empty storage table and call the cycle again.

Do not change stored data manually.

Make a backup of the `TNC:\table\DATA450.KD` file so that you can restore the file, if necessary (e.g. if the data medium is damaged).
19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)

Cycle run
The touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

HEIDENHAIN recommends using the calibration spheres KKH 250 (ID number 655 475-01) or KKH 100 (ID number 655 475-02), which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The TNC evaluates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

1 Clamp the calibration sphere and check for potential collisions.
2 In the Manual Operation mode, set the reference point in the center of the sphere or, if $Q_{431}=1$ or $Q_{431}=3$ is defined: Manually position the touch probe over the calibration sphere in the touch probe axis, and in the center of the sphere in the working plane.
3 Select the Program Run mode and start the calibration program.
4 The TNC automatically measures all three axes successively in the resolution you defined.
5 The TNC saves the measured values in the following Q parameters:
<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>
<tr>
<td>Q147</td>
<td>Offset error in X direction, for manual transfer to the corresponding machine parameter</td>
</tr>
<tr>
<td>Q148</td>
<td>Offset error in Y direction, for manual transfer to the corresponding machine parameter</td>
</tr>
<tr>
<td>Q149</td>
<td>Offset error in Z direction, for manual transfer to the corresponding machine parameter</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°.

Specify the start and end angles to ensure that the same position is not measured twice. A duplicated point measurement (e.g., measuring positions +90° and −270°) is not advisable, but it does not cause an error message.

- Example: Start angle = +90°, end angle = −90°
  - Start angle = +90°
  - End angle = −90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (−90 − +90) / (4 − 1) = −60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = −30°
  - Measuring point 4 = −90°

- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270 − 90) / (4 − 1) = +60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°
Machines with Hirth-coupled axes

Danger of collision!
In order to be positioned, the axis must move out of the Hirth grid. So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere. Also ensure that there is enough space to reach the safety clearance (software limit switch).

Define a retraction height \( \text{Q408} \) greater than 0 if software option 2 (M128, FUNCTION TCPM) is not available.

If necessary, the TNC rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points).

Depending on the machine configuration, the TNC cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine tool builder enabling the TNC to move the rotary axes. The machine tool builder must have entered the number of the M function in machine parameter mStrobeRotAxPos for this purpose.

The measuring positions are calculated from the start angle, end angle and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:
Start angle \( \text{Q411} = -30 \)
End angle \( \text{Q412} = +90 \)
Number of measuring points \( \text{Q414} = 4 \)
Hirth grid = 3°

Calculated stepping angle = \( \frac{\text{Q412} - \text{Q411}}{\text{Q414} - 1} \)

Calculated stepping angle = \( \frac{90 - (-30)}{4 - 1} = \frac{120}{3} = 40 \)

Measuring position 1 = \( \text{Q411} + 0 \times \text{stepping angle} = -30° -> -30° \)
Measuring position 2 = \( \text{Q411} + 1 \times \text{stepping angle} = +10° -> 9° \)
Measuring position 3 = \( \text{Q411} + 2 \times \text{stepping angle} = +50° -> 51° \)
Measuring position 4 = \( \text{Q411} + 3 \times \text{stepping angle} = +90° -> 90° \)
Choice of number of measuring points

To save time you can make a rough optimization with a small number of measuring points (1 or 2), for example during commissioning.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the Check mode.

If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.
Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machine with rotary tables/tilting tables: Clamp the calibrating ball as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

Notes on the accuracy

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error. If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion of results recorded by the TNC in the measuring log is a measure of the machine’s static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.

If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (TRACK column). This generally increases the accuracy of measurements with a 3-D touch probe.

If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine tool manual provides further information.
Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°

- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere on the machine table so that on rotary table axes there is a large measuring circle, or so that on swivel head axes the measurement can be made at a representative position (e.g. in the center of the traverse range).

- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made near the rotary axis angle at which the workpiece is to be machined.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.

- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.

- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is made with the motor encoder, this can result in significant error during tilting.

With input parameter Q432 you can activate backlash measurement. Enter an angle that the TNC uses as traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the TNC will not measure any backlash.

The TNC does not perform an automatic backlash compensation.

- If the measuring circle radius is < 1 mm, the TNC does not calculate the backlash. The larger the measuring circle radius, the more accurately the TNC can determine the rotary axis backlash (see "Logging function", page 637).
- Backlash measurement is not possible if an M function for positioning the rotary axes is set in machine parameter mStrobeRotAxPos or if the axis is a Hirth axis.
Please note while programming:

Note that all functions for tilting in the working plane are reset. M128 and FUNCTION TCPM are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it, or you define the input parameter Q431 correspondingly to 1 or 3.

If machine parameter mStrobeRotAxPos is defined as not equal –1 (M function positions the rotary axis), then only start a measurement when all rotary axes are at 0°.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter Q253 or the FMAX value, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate Q253, while the probe monitoring is inactive.

If the kinematic data attained in the Optimize mode are greater than the permissible limit (maxModification), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

Note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the TNC first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than you have defined in machine parameter maxDevCalBall the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of an emergency the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters. The TNC ignores cycle definition data that applies to inactive axes.
Cycle parameters

- **Mode (0=Check/1=Measure)** Q406: Specify whether the TNC should check or optimize the active kinematics:
  - 0: Check active kinematics. The TNC measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The TNC displays the results of measurement in a measurement log.
  - 1: Optimize active kinematics. The TNC measures the kinematics in the rotary axes you have defined and optimizes the position of the rotary axes of the active kinematics.

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999

- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP in the touch probe table. Input range 0 to 99999.9999; alternatively PREDEF

- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    - Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the sequence A, then B, then C.
  - Input >0:
    - Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. Probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively FMAX, FAUTO, PREDEF

---

### Saving and checking the kinematics

<table>
<thead>
<tr>
<th>Step</th>
<th>Action</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>Tool call “TCH PROBE” Z</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>TCH PROBE 450 SAVE KINEMATICS</td>
<td>Q410=0; MODE; Q409=5; MEMORY DESIGNATION</td>
</tr>
<tr>
<td>6</td>
<td>TCH PROBE 451 MEASURE KINEMATICS</td>
<td>Q406=0; MODE; Q407=12.5; SPHERE RADIUS; Q320=0; SET-UP CLEARANCE; Q408=0; RETR. HEIGHT; Q253=750; F PRE-POSITIONING; Q380=0; REFERENCE ANGLE; Q411=90; START ANGLE A AXIS; Q412=90; END ANGLE A AXIS; Q413=0; INCID. ANGLE A AXIS; Q414=0; MEAS. POINTS A AXIS; Q415=90; START ANGLE B AXIS; Q416=90; END ANGLE B AXIS; Q417=0; INCID. ANGLE B AXIS; Q418=2; MEAS. POINTS B AXIS; Q419=90; START ANGLE C AXIS; Q420=90; END ANGLE C AXIS; Q421=0; INCID. ANGLE C AXIS; Q422=2; MEAS. POINTS C AXIS; Q423=4; NO. OF PROBE POINTS; Q431=0; PRESET; Q432=0; BACKLASH, ANG. RANGE</td>
</tr>
</tbody>
</table>
Reference angle Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000

Start angle A axis Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999

End angle A axis Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999

Angle of incid. A axis Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

Number meas. points A axis Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

Start angle B axis Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999

End angle B axis Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999

Angle of incid. in B axis Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

Number meas. points B axis Q418: Number of probe measurements with which the TNC is to measure the B axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

Start angle C axis Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999

End angle C axis Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999

Angle of incid. in C axis Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999

Number meas. points C axis Q422: Number of probe measurements with which the TNC is to measure the C axis. Input range 0 to 12. If the input value = 0, the TNC does not measure the respective axis.
Number meas. points (3-8) Q423: Number of probe measurements with which the TNC is to measure the calibration sphere in the plane. Input range 3 to 8. Less measuring points increase speed and more measuring points increase measurement precision.

Preset (0/1/2/3) Q431: Define whether the TNC automatically sets the active preset (datum) into the center of the sphere:
- 0: Do not set the preset automatically into the center of the sphere: Preset manually before cycle start
- 1: Automatically preset into the center of the sphere before measurement: Manually preposition the touch probe above the calibration sphere before the cycle start
- 2: Automatically preset into the center of the sphere after measurement: Preset manually before cycle start
- 3: Preset before and after measurement into the center of the sphere: Preposition the touch probe manually above the calibration sphere before cycle start

Backlash, angle range Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range -3.0000 to +3.0000

If you have activated “Preset” before the calibration (Q431 = 1/3), then move the touch probe by the safety clearance (Q320 + SET_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.
Various modes (Q406)

Test mode Q406 = 0
- The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The TNC records the results of a possible position optimization but does not make any adjustments.

Position Optimization mode Q406 = 1
- The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the TNC tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data is adjusted automatically.

Position optimization of the rotary axes with preceding, automatic datum setting and measurement of the rotary axis backlash

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TOOL CALL “TCH PROBE“ Z</td>
</tr>
<tr>
<td>2</td>
<td>T CH PROBE 451 MEASURE KINEMATICS</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q406</td>
<td>1</td>
</tr>
<tr>
<td>Q407</td>
<td>12.5</td>
</tr>
<tr>
<td>Q320</td>
<td>0</td>
</tr>
<tr>
<td>Q408</td>
<td>0</td>
</tr>
<tr>
<td>Q253</td>
<td>750</td>
</tr>
<tr>
<td>Q380</td>
<td>0</td>
</tr>
<tr>
<td>Q411</td>
<td>90</td>
</tr>
<tr>
<td>Q412</td>
<td>90</td>
</tr>
<tr>
<td>Q413</td>
<td>0</td>
</tr>
<tr>
<td>Q414</td>
<td>0</td>
</tr>
<tr>
<td>Q415</td>
<td>90</td>
</tr>
<tr>
<td>Q416</td>
<td>90</td>
</tr>
<tr>
<td>Q417</td>
<td>0</td>
</tr>
<tr>
<td>Q418</td>
<td>0</td>
</tr>
<tr>
<td>Q419</td>
<td>90</td>
</tr>
<tr>
<td>Q420</td>
<td>270</td>
</tr>
<tr>
<td>Q421</td>
<td>0</td>
</tr>
<tr>
<td>Q422</td>
<td>3</td>
</tr>
<tr>
<td>Q423</td>
<td>3</td>
</tr>
<tr>
<td>Q431</td>
<td>1</td>
</tr>
<tr>
<td>Q432</td>
<td>0.5</td>
</tr>
</tbody>
</table>

RANGE
Logging function

After running Cycle 451, the TNC creates a measuring log (TCHPR451.TXT) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes
19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

Cycle run

Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option)", page 624). Then the TNC corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration sphere after optimization.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

1 Clamp the calibration sphere
2 Measure the complete reference head with Cycle 451, and use Cycle 451 to finally set the preset in the center of the sphere.
3 Insert the second head.
4 Use Cycle 452 to measure the interchangeable head up to the point where the head is changed.
5 Use Cycle 452 to adjust other interchangeable heads to the reference head.

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

1 Clamp the calibration sphere and check for potential collisions.
2 Define the preset in the calibration sphere.
3 Set the preset on the workpiece, and start machining the workpiece.
4 Use Cycle 452 for preset compensation at regular intervals. The TNC measures the drift of the axes involved and compensates it in the kinematics description.
<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>
<tr>
<td>Q147</td>
<td>Offset error in X direction, for manual transfer to the corresponding machine parameter</td>
</tr>
<tr>
<td>Q148</td>
<td>Offset error in Y direction, for manual transfer to the corresponding machine parameter</td>
</tr>
<tr>
<td>Q149</td>
<td>Offset error in Z direction, for manual transfer to the corresponding machine parameter</td>
</tr>
</tbody>
</table>
In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

Note that all functions for tilting in the working plane are reset. M128 and FUNCTION TCPM are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it.

For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse a distance of 1° to the limit switch. The TNC needs this distance for internal backlash compensation.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter Q253 or the FMAX value, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate Q253, while the probe monitoring is inactive.

If the kinematic data are greater than the permissible limit (maxModification), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

Note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the TNC first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than you have defined in machine parameter maxDevCalBall the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.
Cycle parameters

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0: Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the sequence A, then B, then C.
  - Input >0: Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. Probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively **FMAX, FAUTO, PREDEF**
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999

Calibration program

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Arguments</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>TOOL CALL “TCH PROBE” Z</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>TCH PROBE 450 SAVE KINEMATICS</td>
<td></td>
</tr>
<tr>
<td>Q410=0 ;MODE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q409=5 ;MEMORY DESIGNATION</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>TCH PROBE 452 PRESET COMPENSATION</td>
<td></td>
</tr>
<tr>
<td>Q407=12.5 ;SPHERE RADIUS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q408=0 ;RETR. HEIGHT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q253=750 ;F PRE-POSITIONING</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q380=0 ;REFERENCE ANGLE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q411=-90 ;START ANGLE A AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q412=+90 ;END ANGLE A AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q413=0 ;INCID. ANGLE A AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q414=0 ;MEAS. POINTS A AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q415=-90 ;START ANGLE B AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q416=+90 ;END ANGLE B AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q417=0 ;INCID. ANGLE B AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q418=2 ;MEAS. POINTS B AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q419=-90 ;START ANGLE C AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q420=+90 ;END ANGLE C AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q421=0 ;INCID. ANGLE C AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q422=2 ;MEAS. POINTS C AXIS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q423=4 ;NO. OF PROBE POINTS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q432=0 ;BACKLASH, ANG. RANGE</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **No. of measuring points** Q423: Specify the number of probing points to be used by the TNC for measuring the calibration sphere in the plane. Input range: 3 to 8 measurements
- **Backlash, angle range** Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range -3.0000 to +3.0000
Adjustment of interchangeable heads
The goal of this procedure is for the workpiece preset to remain unchanged after changing rotary axes (head exchange).
In the following example, a fork head is adjusted to the A and C axes. The A axis is changed, whereas the C axis continues being a part of the basic configuration.
- Insert the interchangeable head that will be used as a reference head.
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle 451 to measure the complete kinematics, including the reference head.
- Set the preset (using Q431 = 2 or 3 in Cycle 451) after measuring the reference head

Measuring a reference head

<table>
<thead>
<tr>
<th>Step</th>
<th>Command</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TOOL CALL &quot;TCH PROBE&quot;</td>
<td>Z</td>
</tr>
<tr>
<td>2</td>
<td>TCH PROBE 451 MEASURE KINEMATICS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q406=1 ;MODE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q407=12.5 ;SPHERE RADIUS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q320=0 ;SET-UP CLEARANCE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q408=0 ;RETR. HEIGHT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q253=2000;F PRE-POSITIONING</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q380=+45 ;REFERENCE ANGLE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q411=90 ;START ANGLE A AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q412=+90 ;END ANGLE A AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q413=45 ;INCID. ANGLE A AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q414=4 ;MEAS. POINTS A AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q415=90 ;START ANGLE B AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q416=+90 ;END ANGLE B AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q417=0 ;INCID. ANGLE B AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q418=2 ;MEAS. POINTS B AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q419=+90 ;START ANGLE C AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q420=+270;END ANGLE C AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q421=0 ;INCID. ANGLE C AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q422=3 ;MEAS. POINTS C AXIS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q423=4 ;NO. OF PROBE POINTS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q431=3 ;PRESET</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Q432=0 ;BACKLASH, ANG. RANGE</td>
<td></td>
</tr>
</tbody>
</table>
19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- Insert the second interchangeable head
- Insert the touch probe
- Measure the interchangeable head with Cycle 452
- Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with Q422)
- The preset and the position of the calibration sphere must not be changed during the complete process
- All other interchangeable heads can be adjusted in the same way

The head change function can vary depending on the individual machine tool. Refer to your machine manual.

<table>
<thead>
<tr>
<th>Adjusting an interchangeable head</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 TOOL CALL &quot;TCH PROBE&quot; Z</td>
</tr>
<tr>
<td>4 TCH PROBE 452 PRESET COMPENSATION</td>
</tr>
<tr>
<td>Q407=12.5 ;SPHERE RADIUS</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q408=0 ;RETR. HEIGHT</td>
</tr>
<tr>
<td>Q253=2000;F PRE-POSITIONING</td>
</tr>
<tr>
<td>Q380=+45 ;REFERENCE ANGLE</td>
</tr>
<tr>
<td>Q411=-90 ;START ANGLE A AXIS</td>
</tr>
<tr>
<td>Q412=+90 ;END ANGLE A AXIS</td>
</tr>
<tr>
<td>Q413=45 ;INCID. ANGLE A AXIS</td>
</tr>
<tr>
<td>Q414=4 ;MEAS. POINTS A AXIS</td>
</tr>
<tr>
<td>Q415=-90 ;START ANGLE B AXIS</td>
</tr>
<tr>
<td>Q416=+90 ;END ANGLE B AXIS</td>
</tr>
<tr>
<td>Q417=0 ;INCID. ANGLE B AXIS</td>
</tr>
<tr>
<td>Q418=2 ;MEAS. POINTS B AXIS</td>
</tr>
<tr>
<td>Q419=+90 ;START ANGLE C AXIS</td>
</tr>
<tr>
<td>Q420=+270;END ANGLE C AXIS</td>
</tr>
<tr>
<td>Q421=0 ;INCID. ANGLE C AXIS</td>
</tr>
<tr>
<td>Q422=0 ;MEAS. POINTS C AXIS</td>
</tr>
<tr>
<td>Q423=4 ;NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q432=0 ;BACKLASH, ANG. RANGE</td>
</tr>
</tbody>
</table>
Drift compensation

During machining various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle 452.

- Clamp the calibration sphere
- Insert the touch probe
- Measure the complete kinematics with Cycle 451 before starting the machining process
- Set the preset (using Q432 = 2 or 3 in Cycle 451) after measuring the kinematics.
- Then set the presets on your workpieces and start the machining process

Reference measurement for drift compensation

<table>
<thead>
<tr>
<th>Step</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TOOL CALL &quot;TCH PROBE&quot; Z</td>
</tr>
<tr>
<td>2</td>
<td>CYCL DEF 247 DATUM SETTING</td>
</tr>
<tr>
<td></td>
<td>Q339=1 ;DATUM NUMBER</td>
</tr>
<tr>
<td>3</td>
<td>TCH PROBE 451 MEASURE KINEMATICS</td>
</tr>
<tr>
<td></td>
<td>Q406=1 ;MODE</td>
</tr>
<tr>
<td></td>
<td>Q407=12.5 ;SPHERE RADIUS</td>
</tr>
<tr>
<td></td>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td></td>
<td>Q408=0 ;RETR. HEIGHT</td>
</tr>
<tr>
<td></td>
<td>Q253=750 ;F PRE-POSITIONING</td>
</tr>
<tr>
<td></td>
<td>Q380=+45 ;REFERENCE ANGLE</td>
</tr>
<tr>
<td></td>
<td>Q411=+90 ;START ANGLE A AXIS</td>
</tr>
<tr>
<td></td>
<td>Q412=+270;END ANGLE A AXIS</td>
</tr>
<tr>
<td></td>
<td>Q413=45 ;INCID. ANGLE A AXIS</td>
</tr>
<tr>
<td></td>
<td>Q414=4 ;MEAS. POINTS A AXIS</td>
</tr>
<tr>
<td></td>
<td>Q415=90 ;START ANGLE B AXIS</td>
</tr>
<tr>
<td></td>
<td>Q416=+90 ;END ANGLE B AXIS</td>
</tr>
<tr>
<td></td>
<td>Q417=0 ;INCID. ANGLE B AXIS</td>
</tr>
<tr>
<td></td>
<td>Q418=2 ;MEAS. POINTS B AXIS</td>
</tr>
<tr>
<td></td>
<td>Q419=+90 ;START ANGLE C AXIS</td>
</tr>
<tr>
<td></td>
<td>Q420=+270;END ANGLE C AXIS</td>
</tr>
<tr>
<td></td>
<td>Q421=0 ;INCID. ANGLE C AXIS</td>
</tr>
<tr>
<td></td>
<td>Q422=3 ;MEAS. POINTS C AXIS</td>
</tr>
<tr>
<td></td>
<td>Q423=4 ;NO. OF PROBE POINTS</td>
</tr>
<tr>
<td></td>
<td>Q431=3 ;PRESET</td>
</tr>
<tr>
<td></td>
<td>Q432=0 ;BACKLASH, ANG. RANGE</td>
</tr>
</tbody>
</table>
Touch Probe Cycles: Automatic Kinematics Measurement

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option)

- Measure the drift of the axes at regular intervals.
- Insert the touch probe
- Activate the preset in the calibration sphere.
- Use Cycle 452 to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the complete process.

This procedure can also be performed on machines without rotary axes.

Drift compensation

<table>
<thead>
<tr>
<th>4 TOOL CALL &quot;TCH PROBE&quot; Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 TCH PROBE 452 PRESET COMPENSATION</td>
</tr>
<tr>
<td>Q407=12.5 ;SPHERE RADIUS</td>
</tr>
<tr>
<td>Q320=0 ;SET-UP CLEARANCE</td>
</tr>
<tr>
<td>Q408=0 ;RETR. HEIGHT</td>
</tr>
<tr>
<td>Q253=99999 ;PRE-POSITIONING</td>
</tr>
<tr>
<td>Q380=45 ;REFERENCE ANGLE</td>
</tr>
<tr>
<td>Q411=-90 ;START ANGLE A AXIS</td>
</tr>
<tr>
<td>Q412=+90 ;END ANGLE A AXIS</td>
</tr>
<tr>
<td>Q413=45 ;INCID. ANGLE A AXIS</td>
</tr>
<tr>
<td>Q414=4 ;MEAS. POINTS A AXIS</td>
</tr>
<tr>
<td>Q415=90 ;START ANGLE B AXIS</td>
</tr>
<tr>
<td>Q416=+90 ;END ANGLE B AXIS</td>
</tr>
<tr>
<td>Q417=0 ;INCID. ANGLE B AXIS</td>
</tr>
<tr>
<td>Q418=2 ;MEAS. POINTS B AXIS</td>
</tr>
<tr>
<td>Q419=+90 ;START ANGLE C AXIS</td>
</tr>
<tr>
<td>Q420=+270 ;END ANGLE C AXIS</td>
</tr>
<tr>
<td>Q421=0 ;INCID. ANGLE C AXIS</td>
</tr>
<tr>
<td>Q422=3 ;MEAS. POINTS C AXIS</td>
</tr>
<tr>
<td>Q423=3 ;NO. OF PROBE POINTS</td>
</tr>
<tr>
<td>Q432=0 ;BACKLASH, ANG. RANGE</td>
</tr>
</tbody>
</table>
Logging function
After running Cycle 452, the TNC creates a measuring log (TCHPR452.TXT) containing the following information:
- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes

Notes on log data
(see "Logging function", page 637)
20

Touch Probe Cycles: Automatic Tool Measurement
20.1 Fundamentals

Overview

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.
Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.
The touch probe cycles are available only with the Touch Probe Functions software option (option number 17). If you are using a HEIDENHAIN touch probe, this option is available automatically.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill
- Tool measurement while the tool is rotating
- Measurement of individual teeth
You can program the cycles for tool measurement in the Programming mode of operation using the TOUCH PROBE key. The following cycles are available:

<table>
<thead>
<tr>
<th>New format</th>
<th>Old format</th>
<th>Cycle</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Calibrating the TT, Cycles 30 and 480</td>
<td>656</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Calibrating the wireless TT 449, Cycle 484</td>
<td>657</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Measuring the tool length, Cycles 31 and 481</td>
<td>659</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Measuring the tool radius, Cycles 32 and 482</td>
<td>661</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Measuring the tool length and radius, Cycles 33 and 483</td>
<td>663</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.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in controls for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter Q199.
20.1 Fundamentals

Setting machine parameters

Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings > CfgToolMeasurement** and **CfgTTRoundStylus**. The TNC uses the feed rate for probing defined in **probingFeed** when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

\[ n = \frac{\text{maxPeriphSpeedMeas}}{r \cdot 0.0063} \]

- **n**: Spindle speed [rpm]
- **maxPeriphSpeedMeas**: Maximum permissible cutting speed in m/min
- **r**: Active tool radius in mm

The feed rate for probing is calculated from:

\[ v = \text{measuring tolerance} \cdot n \]

- **v**: Feed rate for probing in mm/min
- **Measuring tolerance**: Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
- **n**: Shaft speed [rpm]
probingFeedCalc determines the calculation of the probing feed rate:

probingFeedCalc = ConstantTolerance:
The measuring tolerance remains constant regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (maxPeriphSpeedMeas) and the permissible tolerance (measureTolerance1), the sooner you will encounter this effect.

probingFeedCalc = VariableTolerance:
The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

<table>
<thead>
<tr>
<th>Tool radius</th>
<th>Measuring tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Up to 30 mm</td>
<td>measureTolerance1</td>
</tr>
<tr>
<td>30 to 60 mm</td>
<td>2 • measureTolerance1</td>
</tr>
<tr>
<td>60 to 90 mm</td>
<td>3 • measureTolerance1</td>
</tr>
<tr>
<td>90 to 120 mm</td>
<td>4 • measureTolerance1</td>
</tr>
</tbody>
</table>

probingFeedCalc = ConstantFeed:
The feed rate for probing remains constant; the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance = r • measureTolerance1 / 5 mm, where

\( r \): Active tool radius in mm

\( \text{measureTolerance1} \): Maximum permissible error of measurement
## Touch Probe Cycles: Automatic Tool Measurement

### 20.1 Fundamentals

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>Number of teeth?</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>Wear tolerance: length?</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 L). Input range: 0 to 0.9999 mm</td>
<td>Wear tolerance: radius?</td>
</tr>
<tr>
<td><strong>R2TOL</strong></td>
<td>Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm</td>
<td>Wear tolerance: Radius 2?</td>
</tr>
<tr>
<td><strong>DIRECT.</strong></td>
<td>Cutting direction of the tool for measuring the tool during rotation</td>
<td>Cutting direction (M3 = -)?</td>
</tr>
<tr>
<td><strong>R_OFFS</strong></td>
<td>Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)</td>
<td>Tool offset: radius?</td>
</tr>
<tr>
<td><strong>L_OFFS</strong></td>
<td>Tool radius measurement: tool offset in addition to offsetToolAxis between upper surface of stylus and lower surface of tool. Default: 0</td>
<td>Tool offset: length?</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>Breakage tolerance: length?</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 L). Input range: 0 to 0.9999 mm</td>
<td>Breakage tolerance: radius?</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>Drill</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 <code>offsetToolAxis</code> 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 <code>offsetToolAxis</code> is used)</td>
</tr>
<tr>
<td><strong>Radius cutter</strong> with a diameter of 10 mm, for example</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>
Touch Probe Cycles: Automatic Tool Measurement

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

Cycle run
The TT is calibrated with the measuring cycle TCH PROBE 30 or TCH PROBE 480 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", page 651). The calibration process is automatic. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurements.

Please note while programming:

The functioning of the calibration cycle is dependent on machine parameter CfgToolMeasurement. Refer to your machine tool manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The position of the TT within the machine working space must be defined by setting the Machine Parameters centerPos > [0] to [2].

If you change the setting of any of the Machine Parameters centerPos > [0] to [2], you must recalibrate.

Cycle parameters

- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from safetyDistStylus). Input range -9999.9999 to 99999.9999

<table>
<thead>
<tr>
<th>NC blocks in old format</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TOOL CALL 1 Z</td>
</tr>
<tr>
<td>7 TCH PROBE 30.0 CALIBRATE TT</td>
</tr>
<tr>
<td>8 TCH PROBE 30.1 HEIGHT: +90</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>NC blocks in new format</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TOOL CALL 1 Z</td>
</tr>
<tr>
<td>7 TCH PROBE 480 CALIBRATE TT</td>
</tr>
<tr>
<td>Q260=+100;CLEARANCE HEIGHT</td>
</tr>
</tbody>
</table>
20.3 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, DIN/ISO: G484)

Fundamentals
With Cycle 484, you can calibrate your tool touch probe, e.g. the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run
To calibrate the tool touch probe, program the measuring cycle TCH PROBE 484. In the input parameter Q536, you can specify whether you want to run the cycle semi-automatically or fully automatically.

**Semi-automatic—stop before running**
- Insert the calibrating tool
- Define and start the calibration cycle
- The TNC interrupts the calibration cycle
- The TNC opens a dialog in a new window
- The dialog asks you to manually position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact

**Fully automatic—no stop before running**
- Insert the calibrating tool
- Position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact
- Define and start the calibration cycle
- The calibration cycle is executed without stopping. The calibration process starts from the current position of the tool.

Calibrating tool:
The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. Enter the exact length and radius of the calibrating tool into the tool table TOOL.T. At the end of the calibration process, the TNC stores the calibration values and takes them into account during subsequent tool measurement. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.
Touch Probe Cycles: Automatic Tool Measurement

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

Please note while programming:

**Danger of collision!**

To avoid collisions, the tool must be pre-positioned before the cycle call if Q536 is set to 1!

In the calibration process, the TNC also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

The functioning of the calibration cycle is dependent on machine parameter **CfgToolMeasurement**. Refer to your machine manual.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. When using a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. The use of a calibrating tool of too small a diameter and/or protruding too far from the chuck may cause significant inaccuracies.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

**Stop before running** Q536: Specify whether to stop before cycle start or run the cycle automatically without stopping:

0: Stop before running. A dialog asks you to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC start to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process.

1: No stop before running. The TNC starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

**NC blocks**

<table>
<thead>
<tr>
<th>NC blocks</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TOOL CALL 1 Z</td>
</tr>
<tr>
<td>7 TCH PROBE 484 CALIBRATE TT</td>
</tr>
<tr>
<td>Q536=+0 ;STOP BEFORE RUNNING</td>
</tr>
</tbody>
</table>
20.4 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481)

Cycle run
In order to measure the tool length program the measurement cycles TCH PROBE 31 or TCH PROBE 481 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483"). Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is at standstill.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is at standstill.

Cycle for measuring a tool during rotation
The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (TT: R_OFFS).

Cycle for measuring a tool during standstill (e.g. for drills)
The control positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until it touches the surface. To activate this function, enter zero for the tool offset: Radius (TT: R_OFFS) in the tool table.

Cycle for measuring individual teeth
The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in offsetToolAxis. You can enter an additional offset with tool offset: Length (TT: L_OFFS) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for CUTTER MEASUREMENT.
Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction. You can run an individual tooth measurement of tools with up to 20 teeth.

Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q-parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

- **Parameter number for result ?**: Parameter number in which the TNC saves the status of the measurement result:
  - 0.0: Tool is within tolerance
  - 1.0: Tool is worn (LTOL exceeded)
  - 2.0: Tool is broken (LBREAK exceeded)
If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.

- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from safetyDistStylus). Input range -99999.9999 to 99999.9999

- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth)

Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 Q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1

NC blocks in new format

6 TOOL CALL 12 Z
7 TCH PROBE 481 TOOL LENGTH
Q340=1 ;CHECK
Q260=P+100;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH
20.5 Measuring tool radius (Cycle 32 or 482, DIN/ISO: G482)

Cycle run
To measure the tool radius, program the measuring cycle TCH PROBE 32 or TCH PROBE 482 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", page 651). Select via input parameters by which of two methods the radius of a tool is to be measured:
- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in offsetToolAxis. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control measures the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth CUT as 0 and adjust machine parameter CfgToolMeasurement. Refer to your machine manual.
Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius \( R \) in the central tool file TOOL.T by the delta value \( DR = 0 \). If you wish to inspect a tool, the TNC compares the measured radius with the tool radius \( R \) that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q-parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

- **Parameter number for result ?**: Parameter number in which the TNC saves the status of the measurement result:
  - 0.0: Tool is within tolerance
  - 1.0: Tool is worn (RTOL exceeded)
  - 2.0: Tool is broken (RBREAK exceeded).
  If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.

- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from safetyDistStylus). Input range -99999.9999 to 99999.9999.

- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is also to measure the individual teeth (maximum of 20 teeth).

### Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0
```

### Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 Q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1
```

### NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 482 TOOL RADIUS
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH
```
20.6 Measuring tool length and radius
(Cycle 33 or 483, DIN/ISO: G483)

Cycle run
To measure both the length and radius of a tool, program
the measuring cycle TCH PROBE 33 or TCH PROBE 483
(see "Differences between Cycles 31 to 33 and Cycles 481 to
483", page 651). This cycle is particularly suitable for the first
measurement of tools, as it saves time when compared with
individual measurement of length and radius. Via input parameters
you can select the desired type of measurement:
- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently
  measuring the individual teeth.

The TNC measures the tool in a fixed programmed sequence. First
it measures the tool radius, then the tool length. The sequence of
measurement is the same as for Cycles 31 and 32 as well as .

Please note while programming:

Before measuring a tool for the first time, enter the
following data on the tool into the tool table TOOL.T:
the approximate radius, the approximate length, the
number of teeth, and the cutting direction.
Cylindrical tools with diamond surfaces can be
measured with stationary spindle. To do so, define
in the tool table the number of teeth \textit{CUT} as 0 and
adjust machine parameter \textit{CfgToolMeasurement}.
Refer to your machine manual.
Touch Probe Cycles: Automatic Tool Measurement

20.6 Measuring tool length and radius (Cycle 483)

Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R and the tool length L in the central tool file TOOL.T by the delta values DR = 0 and DL = 0. If you wish to inspect a tool, the TNC compares the measured data with the tool data stored in TOOL.T. The TNC calculates the deviations and enters them as positive or negative delta values DR and DL in TOOL.T. The deviations are also available in the Q parameters Q115 and Q116. If the delta values are greater than the permissible tool tolerances for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

- **Parameter number for result ?**: Parameter number in which the TNC saves the status of the measurement result:
  - 0.0: Tool is within tolerance
  - 1.0: Tool is worn (LTOL and/or RTOL exceeded)
  - 2.0: Tool is broken (LBREAK and/or RBREAK exceeded).
  If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.

- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from safetyDistStylus). Input range -99999.9999 to 99999.9999

- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is also to measure the individual teeth (maximum of 20 teeth)

---

Measuring a rotating tool for the first time; old format

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TOOL CALL 12 Z</td>
<td></td>
</tr>
<tr>
<td>7 TCH PROBE 33.0 MEASURE TOOL</td>
<td></td>
</tr>
<tr>
<td>8 TCH PROBE 33.1 CHECK: 0</td>
<td></td>
</tr>
<tr>
<td>9 TCH PROBE 33.2 HEIGHT: +120</td>
<td></td>
</tr>
<tr>
<td>10 TCH PROBE 33.3 PROBING THE TEETH: 0</td>
<td></td>
</tr>
</tbody>
</table>

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TOOL CALL 12 Z</td>
<td></td>
</tr>
<tr>
<td>7 TCH PROBE 33.0 MEASURE TOOL</td>
<td></td>
</tr>
<tr>
<td>8 TCH PROBE 33.1 CHECK: 1 Q5</td>
<td></td>
</tr>
<tr>
<td>9 TCH PROBE 33.2 HEIGHT: +120</td>
<td></td>
</tr>
<tr>
<td>10 TCH PROBE 33.3 PROBING THE TEETH: 1</td>
<td></td>
</tr>
</tbody>
</table>

NC blocks in new format

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TOOL CALL 12 Z</td>
<td></td>
</tr>
<tr>
<td>7 TCH PROBE 483 MEASURE TOOL</td>
<td></td>
</tr>
<tr>
<td>Q340=1 ;CHECK</td>
<td></td>
</tr>
<tr>
<td>Q260=+100 ;CLEARANCE HEIGHT</td>
<td></td>
</tr>
<tr>
<td>Q341=1 ;PROBING THE TEETH</td>
<td></td>
</tr>
</tbody>
</table>
21

Tables of Cycles
# Tables of Cycles

## 21.1 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>263</td>
</tr>
<tr>
<td>8</td>
<td>Mirror image</td>
<td></td>
<td></td>
<td>270</td>
</tr>
<tr>
<td>9</td>
<td>Dwell time</td>
<td></td>
<td></td>
<td>287</td>
</tr>
<tr>
<td>10</td>
<td>Rotation</td>
<td></td>
<td></td>
<td>272</td>
</tr>
<tr>
<td>11</td>
<td>Scaling factor</td>
<td></td>
<td></td>
<td>274</td>
</tr>
<tr>
<td>12</td>
<td>Program call</td>
<td></td>
<td></td>
<td>288</td>
</tr>
<tr>
<td>13</td>
<td>Spindle orientation</td>
<td></td>
<td></td>
<td>290</td>
</tr>
<tr>
<td>14</td>
<td>Contour definition</td>
<td></td>
<td></td>
<td>198</td>
</tr>
<tr>
<td>15</td>
<td>Tilting the working plane</td>
<td></td>
<td></td>
<td>277</td>
</tr>
<tr>
<td>20</td>
<td>Contour data SL II</td>
<td></td>
<td></td>
<td>203</td>
</tr>
<tr>
<td>21</td>
<td>Pilot drilling SL II</td>
<td></td>
<td></td>
<td>205</td>
</tr>
<tr>
<td>22</td>
<td>Rough out SL II</td>
<td></td>
<td></td>
<td>207</td>
</tr>
<tr>
<td>23</td>
<td>Floor finishing SL II</td>
<td></td>
<td></td>
<td>211</td>
</tr>
<tr>
<td>24</td>
<td>Side finishing SL II</td>
<td></td>
<td></td>
<td>213</td>
</tr>
<tr>
<td>25</td>
<td>Contour train</td>
<td></td>
<td></td>
<td>216</td>
</tr>
<tr>
<td>26</td>
<td>Axis-specific scaling</td>
<td></td>
<td></td>
<td>275</td>
</tr>
<tr>
<td>27</td>
<td>Cylinder surface</td>
<td></td>
<td></td>
<td>231</td>
</tr>
<tr>
<td>28</td>
<td>Cylindrical surface slot</td>
<td></td>
<td></td>
<td>234</td>
</tr>
<tr>
<td>29</td>
<td>Cylinder surface ridge</td>
<td></td>
<td></td>
<td>237</td>
</tr>
<tr>
<td>32</td>
<td>Tolerance</td>
<td></td>
<td></td>
<td>291</td>
</tr>
<tr>
<td>39</td>
<td>Cylinder surface contour</td>
<td></td>
<td></td>
<td>240</td>
</tr>
<tr>
<td>200</td>
<td>Drilling</td>
<td></td>
<td></td>
<td>79</td>
</tr>
<tr>
<td>201</td>
<td>Reaming</td>
<td></td>
<td></td>
<td>81</td>
</tr>
<tr>
<td>202</td>
<td>Boring</td>
<td></td>
<td></td>
<td>83</td>
</tr>
<tr>
<td>203</td>
<td>Universal drilling</td>
<td></td>
<td></td>
<td>86</td>
</tr>
<tr>
<td>204</td>
<td>Back boring</td>
<td></td>
<td></td>
<td>89</td>
</tr>
<tr>
<td>205</td>
<td>Universal pecking</td>
<td></td>
<td></td>
<td>92</td>
</tr>
<tr>
<td>206</td>
<td>Tapping with a floating tap holder, new</td>
<td></td>
<td></td>
<td>107</td>
</tr>
<tr>
<td>207</td>
<td>Rigid tapping, new</td>
<td></td>
<td></td>
<td>110</td>
</tr>
<tr>
<td>208</td>
<td>Bore milling</td>
<td></td>
<td></td>
<td>96</td>
</tr>
<tr>
<td>209</td>
<td>Tapping with chip breaking</td>
<td></td>
<td></td>
<td>113</td>
</tr>
<tr>
<td>220</td>
<td>Polar pattern</td>
<td></td>
<td></td>
<td>187</td>
</tr>
<tr>
<td>221</td>
<td>Cartesian pattern</td>
<td></td>
<td></td>
<td>190</td>
</tr>
<tr>
<td>225</td>
<td>Engraving</td>
<td></td>
<td></td>
<td>309</td>
</tr>
<tr>
<td>232</td>
<td>Face milling</td>
<td></td>
<td></td>
<td>313</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>233</td>
<td>Face milling (selectable milling direction, consider the side walls)</td>
<td></td>
<td></td>
<td>174</td>
</tr>
<tr>
<td>239</td>
<td>Ascertain the load</td>
<td></td>
<td></td>
<td>318</td>
</tr>
<tr>
<td>240</td>
<td>Centering</td>
<td></td>
<td></td>
<td>77</td>
</tr>
<tr>
<td>241</td>
<td>Single-lip deep-hole drilling</td>
<td></td>
<td></td>
<td>99</td>
</tr>
<tr>
<td>247</td>
<td>Datum setting</td>
<td></td>
<td></td>
<td>269</td>
</tr>
<tr>
<td>251</td>
<td>Rectangular pocket (complete machining)</td>
<td></td>
<td></td>
<td>143</td>
</tr>
<tr>
<td>252</td>
<td>Circular pocket (complete machining)</td>
<td></td>
<td></td>
<td>147</td>
</tr>
<tr>
<td>253</td>
<td>Slot milling</td>
<td></td>
<td></td>
<td>152</td>
</tr>
<tr>
<td>254</td>
<td>Circular slot</td>
<td></td>
<td></td>
<td>156</td>
</tr>
<tr>
<td>256</td>
<td>Rectangular stud (complete machining)</td>
<td></td>
<td></td>
<td>161</td>
</tr>
<tr>
<td>257</td>
<td>Circular stud (complete machining)</td>
<td></td>
<td></td>
<td>165</td>
</tr>
<tr>
<td>258</td>
<td>Polygon stud</td>
<td></td>
<td></td>
<td>169</td>
</tr>
<tr>
<td>262</td>
<td>Thread milling</td>
<td></td>
<td></td>
<td>119</td>
</tr>
<tr>
<td>263</td>
<td>Thread milling/countersinking</td>
<td></td>
<td></td>
<td>122</td>
</tr>
<tr>
<td>264</td>
<td>Thread drilling/milling</td>
<td></td>
<td></td>
<td>126</td>
</tr>
<tr>
<td>265</td>
<td>Helical thread drilling/milling</td>
<td></td>
<td></td>
<td>130</td>
</tr>
<tr>
<td>267</td>
<td>Outside thread milling</td>
<td></td>
<td></td>
<td>134</td>
</tr>
<tr>
<td>270</td>
<td>Contour train data</td>
<td></td>
<td></td>
<td>218</td>
</tr>
<tr>
<td>275</td>
<td>Trochoidal slot</td>
<td></td>
<td></td>
<td>219</td>
</tr>
<tr>
<td>291</td>
<td>Coupling turning interpolation</td>
<td></td>
<td></td>
<td>294</td>
</tr>
<tr>
<td>292</td>
<td>Contour turning interpolation</td>
<td></td>
<td></td>
<td>303</td>
</tr>
</tbody>
</table>
# Tables of Cycles

## 21.1 Overview

### Turning 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>800</td>
<td>Adapt rotary coordinate system</td>
<td></td>
<td></td>
<td>332</td>
</tr>
<tr>
<td>801</td>
<td>Reset rotary coordinate system</td>
<td></td>
<td></td>
<td>338</td>
</tr>
<tr>
<td>810</td>
<td>Turn contour, longitudinal</td>
<td></td>
<td></td>
<td>354</td>
</tr>
<tr>
<td>811</td>
<td>Turn shoulder, longitudinal</td>
<td></td>
<td></td>
<td>340</td>
</tr>
<tr>
<td>812</td>
<td>Turn shoulder, longitudinal extended</td>
<td></td>
<td></td>
<td>343</td>
</tr>
<tr>
<td>813</td>
<td>Turn, longitudinal plunge</td>
<td></td>
<td></td>
<td>347</td>
</tr>
<tr>
<td>814</td>
<td>Turn, longitudinal plunge extended</td>
<td></td>
<td></td>
<td>350</td>
</tr>
<tr>
<td>815</td>
<td>Turn contour-parallel</td>
<td></td>
<td></td>
<td>358</td>
</tr>
<tr>
<td>820</td>
<td>Turn contour, transverse</td>
<td></td>
<td></td>
<td>376</td>
</tr>
<tr>
<td>821</td>
<td>Turn shoulder face</td>
<td></td>
<td></td>
<td>362</td>
</tr>
<tr>
<td>822</td>
<td>Turn shoulder face extended</td>
<td></td>
<td></td>
<td>365</td>
</tr>
<tr>
<td>823</td>
<td>Turn, transverse plunge</td>
<td></td>
<td></td>
<td>369</td>
</tr>
<tr>
<td>824</td>
<td>Turn, transverse plunge extended</td>
<td></td>
<td></td>
<td>372</td>
</tr>
<tr>
<td>830</td>
<td>Thread, contour-parallel</td>
<td></td>
<td></td>
<td>431</td>
</tr>
<tr>
<td>831</td>
<td>Thread, longitudinal</td>
<td></td>
<td></td>
<td>424</td>
</tr>
<tr>
<td>832</td>
<td>Thread, extended</td>
<td></td>
<td></td>
<td>427</td>
</tr>
<tr>
<td>860</td>
<td>Recessing contour, radial</td>
<td></td>
<td></td>
<td>411</td>
</tr>
<tr>
<td>861</td>
<td>Recessing, radial</td>
<td></td>
<td></td>
<td>404</td>
</tr>
<tr>
<td>862</td>
<td>Recessing, radial extended</td>
<td></td>
<td></td>
<td>407</td>
</tr>
<tr>
<td>870</td>
<td>Recessing contour, axial</td>
<td></td>
<td></td>
<td>421</td>
</tr>
<tr>
<td>871</td>
<td>Recessing, axial</td>
<td></td>
<td></td>
<td>415</td>
</tr>
<tr>
<td>872</td>
<td>Recessing, axial extended</td>
<td></td>
<td></td>
<td>417</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>540</td>
</tr>
<tr>
<td>1</td>
<td>Polar datum</td>
<td>■</td>
<td></td>
<td>541</td>
</tr>
<tr>
<td>3</td>
<td>Measuring</td>
<td>■</td>
<td></td>
<td>581</td>
</tr>
<tr>
<td>4</td>
<td>Measuring in 3-D</td>
<td>■</td>
<td></td>
<td>583</td>
</tr>
<tr>
<td>30</td>
<td>Calibrate the TT</td>
<td>■</td>
<td></td>
<td>656</td>
</tr>
<tr>
<td>31</td>
<td>Measure/Inspect the tool length</td>
<td>■</td>
<td></td>
<td>659</td>
</tr>
<tr>
<td>32</td>
<td>Measure/inspect the tool radius</td>
<td>■</td>
<td></td>
<td>661</td>
</tr>
<tr>
<td>33</td>
<td>Measure/Inspect the tool length and the tool radius</td>
<td>■</td>
<td></td>
<td>663</td>
</tr>
<tr>
<td>400</td>
<td>Basic rotation using two points</td>
<td>■</td>
<td></td>
<td>460</td>
</tr>
<tr>
<td>401</td>
<td>Basic rotation over two holes</td>
<td>■</td>
<td></td>
<td>463</td>
</tr>
<tr>
<td>402</td>
<td>Basic rotation over two studs</td>
<td>■</td>
<td></td>
<td>466</td>
</tr>
<tr>
<td>403</td>
<td>Compensate misalignment with rotary axis</td>
<td>■</td>
<td></td>
<td>469</td>
</tr>
<tr>
<td>404</td>
<td>Set basic rotation</td>
<td>■</td>
<td></td>
<td>472</td>
</tr>
<tr>
<td>405</td>
<td>Compensate misalignment with the C axis</td>
<td>■</td>
<td></td>
<td>473</td>
</tr>
<tr>
<td>408</td>
<td>Reference point at slot center (FCL 3 function)</td>
<td>■</td>
<td></td>
<td>484</td>
</tr>
<tr>
<td>409</td>
<td>Reference point at ridge center (FCL 3 function)</td>
<td>■</td>
<td></td>
<td>488</td>
</tr>
<tr>
<td>410</td>
<td>Datum from inside of rectangle</td>
<td>■</td>
<td></td>
<td>491</td>
</tr>
<tr>
<td>411</td>
<td>Datum from outside of rectangle</td>
<td>■</td>
<td></td>
<td>495</td>
</tr>
<tr>
<td>412</td>
<td>Datum from inside of circle (hole)</td>
<td>■</td>
<td></td>
<td>498</td>
</tr>
<tr>
<td>413</td>
<td>Datum from outside of circle (stud)</td>
<td>■</td>
<td></td>
<td>503</td>
</tr>
<tr>
<td>414</td>
<td>Datum from outside of corner</td>
<td>■</td>
<td></td>
<td>507</td>
</tr>
<tr>
<td>415</td>
<td>Datum from inside of corner</td>
<td>■</td>
<td></td>
<td>512</td>
</tr>
<tr>
<td>416</td>
<td>Datum from circle center</td>
<td>■</td>
<td></td>
<td>516</td>
</tr>
<tr>
<td>417</td>
<td>Datum in touch probe axis</td>
<td>■</td>
<td></td>
<td>520</td>
</tr>
<tr>
<td>418</td>
<td>Datum at center between four holes</td>
<td>■</td>
<td></td>
<td>522</td>
</tr>
<tr>
<td>419</td>
<td>Datum in any one axis</td>
<td>■</td>
<td></td>
<td>526</td>
</tr>
<tr>
<td>420</td>
<td>Workpiece—measure angle</td>
<td>■</td>
<td></td>
<td>542</td>
</tr>
<tr>
<td>421</td>
<td>Workpiece—measure hole (center and diameter of hole)</td>
<td>■</td>
<td></td>
<td>545</td>
</tr>
<tr>
<td>422</td>
<td>Workpiece—measure circle from outside (diameter of circular stud)</td>
<td>■</td>
<td>550</td>
<td></td>
</tr>
<tr>
<td>423</td>
<td>Workpiece—measure rectangle from inside</td>
<td>■</td>
<td></td>
<td>555</td>
</tr>
<tr>
<td>424</td>
<td>Workpiece—measure rectangle from outside</td>
<td>■</td>
<td></td>
<td>558</td>
</tr>
<tr>
<td>425</td>
<td>Workpiece—measure inside width (slot)</td>
<td>■</td>
<td></td>
<td>561</td>
</tr>
<tr>
<td>426</td>
<td>Workpiece—measure outside width (ridge)</td>
<td>■</td>
<td></td>
<td>564</td>
</tr>
<tr>
<td>427</td>
<td>Workpiece—measure in any selectable axis</td>
<td>■</td>
<td></td>
<td>567</td>
</tr>
<tr>
<td>430</td>
<td>Workpiece—measure bolt hole circle</td>
<td>■</td>
<td></td>
<td>570</td>
</tr>
<tr>
<td>431</td>
<td>Workpiece—measure plane</td>
<td>■</td>
<td></td>
<td>570</td>
</tr>
</tbody>
</table>
# Tables of Cycles

## 21.1 Overview

<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>450</td>
<td>KinematicsOpt: Save kinematics (option)</td>
<td>□</td>
<td></td>
<td>621</td>
</tr>
<tr>
<td>451</td>
<td>KinematicsOpt: Measure kinematics (option)</td>
<td>□</td>
<td></td>
<td>624</td>
</tr>
<tr>
<td>452</td>
<td>KinematicsOpt: Preset compensation</td>
<td>□</td>
<td></td>
<td>618</td>
</tr>
<tr>
<td>460</td>
<td>Calibrate the touch probe</td>
<td>□</td>
<td></td>
<td>587</td>
</tr>
<tr>
<td>461</td>
<td>Calibrate touch probe length</td>
<td>□</td>
<td></td>
<td>589</td>
</tr>
<tr>
<td>462</td>
<td>Calibrate touch probe inside radius</td>
<td>□</td>
<td></td>
<td>591</td>
</tr>
<tr>
<td>463</td>
<td>Calibrate touch probe outside radius</td>
<td>□</td>
<td></td>
<td>593</td>
</tr>
<tr>
<td>480</td>
<td>Calibrate the TT</td>
<td>□</td>
<td></td>
<td>656</td>
</tr>
<tr>
<td>481</td>
<td>Measure/Inspect the tool length</td>
<td>□</td>
<td></td>
<td>659</td>
</tr>
<tr>
<td>482</td>
<td>Measure/Inspect the tool radius</td>
<td>□</td>
<td></td>
<td>661</td>
</tr>
<tr>
<td>483</td>
<td>Measure/Inspect the tool length and the tool radius</td>
<td>□</td>
<td></td>
<td>663</td>
</tr>
<tr>
<td>484</td>
<td>Calibrate TT</td>
<td>□</td>
<td></td>
<td>657</td>
</tr>
<tr>
<td>600</td>
<td>Global workspace</td>
<td>□</td>
<td></td>
<td>606</td>
</tr>
<tr>
<td>601</td>
<td>Local workspace</td>
<td>□</td>
<td></td>
<td>611</td>
</tr>
</tbody>
</table>
Index

B
Back boring.......................... 89
Basic rotation
  Measure during program run... 458
Blank form update................ 330
Bolt hole circle................... 187
Bore milling......................... 96
Boring.................................. 83

C
Centering................................ 77
Circular pocket
  Roughing+finishing............... 147
Circular point patterns........... 187
Circular slot
  Roughing+finishing............... 156
Circular stud......................... 165, 169
Classification of results.......... 537
Compensating workpiece
  misalignment....................... 458
  By measuring two points on a
  straight line........................ 460
Over two circular studs.......... 466
Over two holes...................... 463
Via rotary axis..................... 469, 473
Confidence interval................ 453
Consideration of basic rotation.. 448
Contour cycles...................... 196
Contour train........................ 216, 218
CONTOUR TURNING INTERPOLATION... 294
Coordinate transformation........ 262
COUPLING INTERPOLATION
  TURNING................................ 303
  Cycle.................................. 56
  Calling................................ 58
  Define.................................. 57
  Cycles and point tables........... 73
  Cylinder surface
    Machine contour.................. 231, 240
    Ridge machining.................. 237
    Slot machining................... 234

D
Datum shift.......................... 263
  In the program...................... 263
With datum tables.................. 264
Drilling............................... 79, 86, 92
Drilling Cycles...................... 76
Dwell time............................ 287

E
Engraving................................ 309
Face milling.......................... 313
FCL function.......................... 9
Feature Content Level.............. 9
Floor finishing....................... 211
FUNCTION TURNDATA................... 330
Fundamentals of Thread
  Milling................................ 117

G
GEAR HOBBING......................... 435, 440

H
Helical thread drilling/milling.... 130
Inside thread milling............ 119

I
Inside thread milling............ 119

K
Kinematic measurement
  Accuracy.............................. 629
  Backlash............................ 631
  Calibration methods............... 630, 643, 645
  Hirth coupling........................ 627
  Logging function................. 622, 637, 647
  Measuring point selection........ 623, 628
  Measuring position selection.... 629
  Prerequisites ....................... 620
Kinematics measurement........... 618
Kinematics Measurement
  Measure kinematics............... 624, 638
  Kinematics measurement........... 621
  Save kinematics.................... 618
  KinematicsOpt....................... 618

L
Linear point patterns............... 190

M
Machine parameters for 3D touch
  probe................................ 451
Machining pattern.................. 64
Measure angle........................ 542
Measure angle of a plane........ 573
Measure any coordinate........... 567
Measure bolt hole circle.......... 570
Measure hole.......................... 545
Measure hole inside................. 545
Measure hole outside.............. 550
Measure kinematics.................. 624
Preset compensation................. 638
Measurement parameters............ 537
Measurement results in Q
  parameters......................... 537
Measure rectangular pocket...... 558
Measure rectangular stud......... 555
Measure the plane angle........... 573
Measure the slot width............ 561
Measure the width of a
  ridge.................................. 564, 564
Measuring slot width............... 561
Measuring the width of a ridge... 564
Mirroring............................... 270
Multiple measurements............... 453

O
Outside thread milling............. 134

P
Pattern definition.................... 64
Peck drilling.......................... 92, 99
Point patterns......................... 186
Overview................................ 186
Point tables............................ 71
Positioning logic...................... 454
Probing feed rate..................... 452
Program call........................... 288
Via cycle............................... 288

R
Reaming.................................. 81
Recording measurement results.... 535
Rectangular pocket
  Roughing+finishing............... 143
Rectangular stud..................... 161
Reference images..................... 596
Reset rotary coordinate system...... 338
Rotation.................................. 272
Roughing:See SL Cycles,
  Roughing.............................. 207
Index

S

Scaling........................................ 274
Set a basic rotation.................. 472
Side finishing.......................... 213
Single-lip deep-hole drilling........ 99
SL Cycles.................. 196, 231, 240
  Contour cycle.......................... 198
  Contour data................................ 203
  Contour train.............................. 216, 218
Floor finishing........................... 211
SL cycles
  Fundamentals.......................... 196
  Fundamentals............................ 258
SL Cycles
  Pilot drilling.......................... 205
  Roughing................................... 207
  Side finishing............................ 213
  Superimposed contours. 199, 252
SL cycles with complex contour formula........................................ 248
SL cycles with simple contour formula........................................ 258
Slot milling
  Roughing+finishing....................... 152
Spindle orientation...................... 290

T

Tapping
  With a floating tap holder........ 107
  With chip breaking..................... 113
  Without a floating tap holder........ 110, 113
Thread drilling/milling............. 126
Thread milling/countersinking...... 122
Tilting function
  Procedure.................................. 282
  Tilting the working plane.. 277, 277
  Cycle........................................ 277
Tolerance monitoring.................. 537
Tool compensation...................... 538
Tool measurement...................... 650, 654
  Calibrate TT.............................. 656, 657
Machine parameters................... 652
Measuring tool length and radius........................................ 663
Tool length.................................. 659
Tool radius.................................. 661
Tool monitoring.......................... 538
Touch probe cycles
  For automatic mode..................... 450
  Touch probe data.......................... 456
  Touch probe table........................ 455
  Turning cycles............................ 326, 339
  Axial recessing........................ 392, 400, 415, 421
  Contour face............................... 376
  Contour longitudinal.......................... 354
  Contour-parallel............................ 358
Radial recessing...................... 380, 404
  Recessing, axial extended..... 395, 417
  Recessing, radial extended...... 383, 407
  Recessing contour, radial.... 388, 411
  Shoulder, longitudinal extended.... 343
Shoulder face............................... 362
Shoulder face extended................. 365
Shoulder longitudinal................... 340
  Thread, contour-parallel................. 431
  Thread, extended............................ 427
  Thread, longitudinal..................... 424
  Transverse plunge...................... 369
  Transverse plunge extended........ 372
  Turn, longitudinal plunge............ 347
  Turn, longitudinal plunge extended.......................... 350

U

Universal drilling..................... 86, 92

W

Workpiece Measurement............. 534
Touch probes from HEIDENHAIN help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

**Workpiece touch probes**
- **TS 220** Signal transmission by cable
- **TS 440, TS 444** Infrared transmission
- **TS 640, TS 740** Infrared transmission
  - Workpiece alignment
  - Setting datums
  - Workpiece measurement

**Tool touch probes**
- **TT 140** Signal transmission by cable
- **TT 449** Infrared transmission
- **TL** Contact-free laser systems
  - Tool measurement
  - Wear monitoring
  - Tool breakage detection