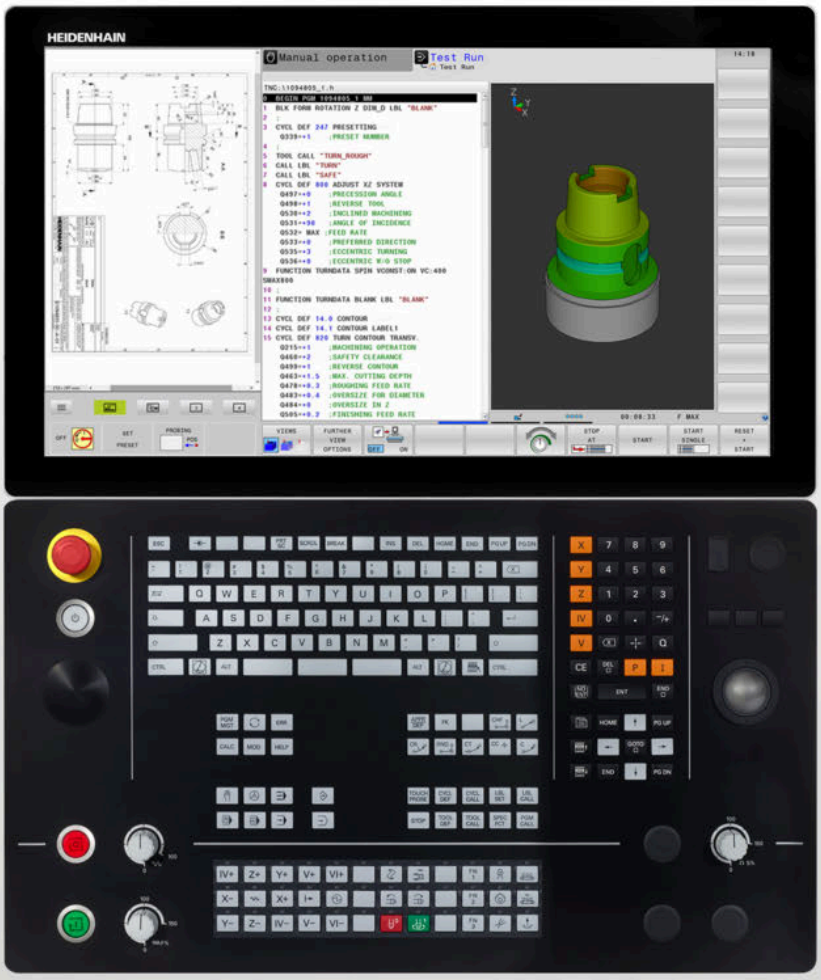
HEIDENHAIN



TNC 640

User's Manual for
Cycle Programming

NC Software
340590-10
340591-10
340595-10

English (en)
10/2019

Contents

1 Fundamentals....43
2 Fundamentals / Overviews....57
3 Using Fixed Cycles....61
4 Fixed Cycles: Drilling....83
5 Fixed Cycles: Tapping / Thread Milling....127
6 Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling....165
7 Cycles: Coordinate Transformations....217
8 Fixed Cycles: Pattern definitions....243
9 Fixed Cycles: Contour Pocket....257
10 Machining Cycles: Optimized contour milling....303
11 Fixed Cycles: Cylindrical Surface....321
12 Fixed Cycles: Contour Pocket with Contour Formula....339
13 Cycles: Special Functions....355
14 Cycles: Turning....425
15 Cycles: Grinding....561
16 Using Touch Probe Cycles....589
17 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment....599
18 Touch Probe Cycles: Automatic Presetting....647
19 Touch Probe Cycles: Automatic Workpiece Inspection....705
20 Touch Probe Cycles: Special Functions....753
21 VSC: camera-based setup control (option 136)....781
22 Touch Probe Cycles: Automatic Kinematics Measurement....801
23 Touch Probe Cycles: Automatic Tool Measurement....841
24 Tables of Cycles....865

1 Fundamentals 43

1.1 About this manual 44

1.2 Control model, software, and features 46

Software options 47

1.3 New or changed cycle functions of software 34059x-09 53

1.4 New or changed cycle functions of software 34059x-10 55

2 Fundamentals / Overviews 57

2.1 Introduction 58

2.2 Available cycle groups 59

Overview of fixed cycles 59

Overview of touch probe cycles 60

3 Using Fixed Cycles 61

3.1 Working with fixed cycles 62

Machine-specific cycles 62
Defining a cycle using soft keys 63
Defining a cycle using the GOTO function 63
Calling a cycle 64
Working with a parallel axis 67

3.2 Program defaults for cycles 68

Overview 68
Entering GLOBAL DEF 68
Using GLOBAL DEF information 69
Global data valid everywhere 70
Global data for drilling operations 70
Global data for milling operations with pocket cycles 25x 70
Global data for milling operations with contour cycles 70
Global data for positioning behavior 71
Global data for probing functions 71

3.3 Pattern definition with PATTERN DEF 72

Application 72
Entering PATTERN DEF 73
Using PATTERN DEF 73
Defining individual machining positions 74
Defining a single row 74
Defining an individual pattern 75
Defining an individual frame 76
Defining a full circle 77
Defining a pitch circle 78

3.4 Point tables 79

Application 79
Entering values into a point table 79
Hiding single points from the machining process 80
Selecting a point table in the NC program 80
Calling a cycle in connection with point tables 81

4 Fixed Cycles: Drilling 83

4.1 Fundamentals 84

Overview 84

4.2 DRILLING (Cycle 200, DIN/ISO: G200) 85

Cycle run 85
Please note while programming: 85
Cycle parameters 86

4.3 REAMING (Cycle 201,DIN/ISO: G201) 87

Cycle run 87
Please note while programming: 87
Cycle parameters 88

4.4 BORING (Cycle 202, DIN/ISO: G202) 89

Cycle run 89
Please note while programming: 90
Cycle parameters 91

4.5 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203) 92

Cycle run 92
Please note while programming: 95
Cycle parameters 96

4.6 BACK BORING (Cycle 204, DIN/ISO: G204) 98

Cycle run 98
Please note while programming: 99
Cycle parameters 100

4.7 UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205) 102

Cycle run 102
Please note while programming: 103
Cycle parameters 104
Position behavior when working with Q379 106

4.8 BORE MILLING (Cycle 208, DIN/ISO: G208) 110

Cycle run 110
Please note while programming: 111
Cycle parameters 112

4.9 SINGLE-LIP DEEP HOLE DRILLING (Cycle 241, DIN/ISO: G241) 113

Cycle run 113
Please note while programming: 114
Cycle parameters 115
Position behavior when working with Q379 117

4.10 CENTERING (Cycle 240, DIN/ISO: G240)..... 121

Cycle run..... 121
Please note while programming:..... 121
Cycle parameters..... 122

4.11 Programming Examples..... 123

Example: Drilling cycles..... 123
Example: Using drilling cycles in connection with PATTERN DEF..... 124

5 Fixed Cycles: Tapping / Thread Milling 127

5.1 Fundamentals 128

Overview 128

5.2 TAPPING with a floating tap holder (Cycle 206, ISO: G206) 129

Cycle run 129
Please note while programming: 130
Cycle parameters 131

5.3 TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207) 132

Cycle run 132
Please note while programming! 132
Cycle parameters 134
Retracting after a program interruption 135

5.4 TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209) 136

Cycle run 136
Please note while programming: 137
Cycle parameters 139
Retracting after a program interruption 140

5.5 Fundamentals of Thread Milling 141

Prerequisites 141

5.6 THREAD MILLING (Cycle 262, DIN/ISO: G262) 143

Cycle run 143
Please note while programming: 144
Cycle parameters 145

5.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263) 147

Cycle run 147
Please note while programming: 148
Cycle parameters 149

5.8 THREAD MILLING (Cycle 264, DIN/ISO: G264) 151

Cycle run 151
Please note while programming: 152
Cycle parameters 153

5.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265) 155

Cycle run 155
Please note while programming: 156
Cycle parameters 157

5.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267) 159

Cycle run 159

Please note while programming:.....160
Cycle parameters.....161

5.11 Programming Examples.....163
Example: Thread milling.....163

6 Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling 165

6.1 Fundamentals 166

Overview 166

6.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251) 167

Cycle run 167
Please note while programming! 168
Cycle parameters 170

6.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252) 173

Cycle run 173
Please note while programming: 175
Cycle parameters 177

6.4 SLOT MILLING (Cycle 253, DIN/ISO: G253) 180

Cycle run 180
Please note while programming: 181
Cycle parameters 182

6.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254) 185

Cycle run 185
Please note while programming: 186
Cycle parameters 188

6.6 RECTANGULAR STUD (Cycle 256, DIN/ISO: G256) 191

Cycle run 191
Please note while programming: 192
Cycle parameters 193

6.7 CIRCULAR STUD (Cycle 257, DIN/ISO: G257) 196

Cycle run 196
Please note while programming: 197
Cycle parameters 198

6.8 POLYGON STUD (Cycle 258, DIN/ISO: G258) 200

Cycle run 200
Please note while programming: 201
Cycle parameters 203

6.9 FACE MILLING (Cycle 233, DIN/ISO: G233) 206

Cycle run 206
Please note while programming: 210
Cycle parameters 211

6.10 Programming Examples 214

Example: Milling pockets, studs and slots 214

7 Cycles: Coordinate Transformations 217

7.1 Fundamentals 218

Overview 218
Effectiveness of coordinate transformations 218

7.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54) 219

Effect 219
Please note while programming 219
Cycle parameters 219

7.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53) 220

Effect 220
Please note while programming: 221
Cycle parameters 221
Selecting a datum table in the part program 222
Editing the datum table in the Programming mode of operation 222
Editing the datum table in the Program Run, Single Block or Program Run, Full Sequence operating mode 224
Configuring a datum table 224
Leaving a datum table 225
Status displays 225

7.4 MIRRORING (Cycle 8, DIN/ISO: G28) 226

Effect 226
Please note while programming: 227
Cycle parameters 227

7.5 ROTATION (Cycle 10, DIN/ISO: G73) 228

Effect 228
Please note while programming: 229
Cycle parameters 229

7.6 SCALING (Cycle 11, DIN/ISO: G72) 230

Effect 230
Cycle parameters 230

7.7 AXIS-SPECIFIC SCALING (Cycle 26) 231

Effect 231
Please note while programming: 231
Cycle parameters 232

7.8 WORKING PLANE (Cycle 19, DIN/ISO: G80, option 1) 233

Effect 233
Please note while programming: 234
Cycle parameters 235
Reset 236

Positioning the axes of rotation....236
Position display in a tilted system....237
Monitoring of the working space....237
Positioning in a tilted coordinate system....238
Combining coordinate transformation cycles....238
Procedure for working with Cycle 19 WORKING PLANE....239

7.9 PRESETTING (Cycle 247, DIN/ISO: G247)....240

Effect....240
Please note before programming:....240
Cycle parameters....240
Status displays....240

7.10 Programming Examples....241

Example: coordinate transformation cycles....241

8 Fixed Cycles: Pattern definitions 243

8.1 Fundamentals 244

Overview 244

8.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220) 246

Cycle run 246
Please note while programming: 246
Cycle parameters 247

8.3 LINEAR POINT PATTERN (Cycle 221, DIN/ISO: G221) 249

Cycle run 249
Please note while programming: 249
Cycle parameters 250

8.4 DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224) 251

Cycle run 251
Please note while programming! 252
Cycle parameters 253

8.5 Programming Examples 254

Example: Polar hole patterns 254

9 Fixed Cycles: Contour Pocket 257

9.1 SL Cycles 258

Fundamentals 258
Overview 260

9.2 CONTOUR (Cycle 14, DIN/ISO: G37) 261

Please note while programming: 261
Cycle parameters 261

9.3 Superimposed contours 262

Fundamentals 262
Subprograms: overlapping pockets 262
Area of inclusion 263
Area of exclusion 264
Area of intersection 265

9.4 CONTOUR DATA (Cycle 20, DIN/ISO: G120) 266

Please note while programming: 266
Cycle parameters 267

9.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121) 268

Cycle run 268
Please note while programming: 269
Cycle parameters 269

9.6 ROUGHING (Cycle 22, DIN/ISO: G122) 270

Cycle run 270
Please note while programming: 271
Cycle parameters 273

9.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123) 275

Cycle run 275
Please note while programming: 276
Cycle parameters 276

9.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124) 277

Cycle run 277
Please note while programming: 278
Cycle parameters 279

9.9 CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270) 280

Please note while programming: 280
Cycle parameters 280

9.10 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125) 281

Cycle run 281

Please note while programming! ... 282
Cycle parameters ... 283

9.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275) ... 285

Cycle run ... 285
Please note while programming: ... 287
Cycle parameters ... 288

9.12 THREE-D CONT. TRAIN (Cycle 276, DIN/ISO: G276) ... 291

Cycle run ... 291
Please note while programming: ... 292
Cycle parameters ... 294

9.13 Programming Examples ... 296

Example: Roughing-out and fine-roughing a pocket ... 296
Example: Pilot drilling, roughing-out and finishing overlapping contours ... 298
Example: Contour train ... 300

10 Machining Cycles: Optimized contour milling 303

10.1 OCM cycles (option 167) 304

OCM fundamentals 304
Overview 306

10.2 OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167) 307

Cycle run 307
Please note while programming! 307
Cycle parameters 308

10.3 OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167) 309

Cycle run 309
Please note while programming! 309
Cycle parameters 310

10.4 OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167) 312

Cycle run 312
Please note while programming! 312
Cycle parameters 313

10.5 OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167) 314

Cycle run 314
Please note while programming! 314
Cycle parameters 315

10.6 Programming examples 316

Example: Open pocket and fine roughing with OCM cycles 316
Example: Program various depths with OCM cycles 319

11 Fixed Cycles: Cylindrical Surface 321

11.1 Fundamentals 322

Overview of cylindrical surface cycles 322

11.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, option 1) 323

Cycle run 323
Please note while programming: 324
Cycle parameters 325

11.3 CYLINDER SURFACE slot milling (Cycle 28, DIN/ISO: G128, option 1) 326

Cycle run 326
Please note while programming: 327
Cycle parameters 329

11.4 CYLINDRICAL SURFACE ridge milling (Cycle 29, DIN/ISO: G129, option 1) 330

Cycle run 330
Please note while programming: 331
Cycle parameters 332

11.5 CYLINDRICAL SURFACE CONTOUR (Cycle 39, DIN/ISO: G139, option 1) 333

Cycle run 333
Please note while programming: 334
Cycle parameters 335

11.6 Programming Examples 336

Example: Cylinder surface with Cycle 27 336
Example: Cylinder surface with Cycle 28 338

12 Fixed Cycles: Contour Pocket with Contour Formula....339

12.1 SL cycles with complex contour formula....340

Fundamentals....340
Selecting an NC program with contour definitions....342
Defining contour descriptions....343
Entering a complex contour formula....344
Superimposed contours....345
Contour machining with SL Cycles....347
Example: Roughing and finishing superimposed contours with the contour formula....348

12.2 SL cycles with simple contour formula....351

Fundamentals....351
Entering a simple contour formula....353
Contour machining with SL Cycles....354

13 Cycles: Special Functions 355

13.1 Fundamentals 356

Overview 356

13.2 DWELL TIME (Cycle 9, ISO: G04) 357

Function 357
Cycle parameters 357

13.3 PROGRAM CALL (Cycle 12, ISO: G39) 358

Cycle function 358
Please note while programming: 358
Cycle parameters 358

13.4 SPINDLE ORIENTATION (Cycle 13, ISO: G36) 359

Cycle function 359
Please note while programming: 359
Cycle parameters 359

13.5 TOLERANCE (Cycle 32, DIN/ISO: G62) 360

Cycle function 360
Influences of the geometry definition in the CAM system 360
Please note while programming! 361
Cycle parameters 363

13.6 COUPLING TURNING INTERPOLATION (Cycle 291, DIN/ISO: G291, option 96) 364

Cycle run 364
Please note while programming! 365
Cycle parameters 366
Defining the tool 367

13.7 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ ISO : G292, option 96) 371

Cycle run 371
Please note while programming: 373
Cycle parameters 375
Machining variants 377
Defining the tool 379

13.8 ENGRAVING (Cycle 225, ISO: G225) 382

Cycle run 382
Please note while programming: 382
Cycle parameters 383
Allowed engraving characters 385
Characters that cannot be printed 385
Engraving system variables 386
Engraving the name and path of an NC program 387
Engraving the counter reading 387

13.9 FACE MILLING (Cycle 232, ISO: G232)..... 388
Cycle run..... 388
Please note while programming:..... 390
Cycle parameters..... 391

13.10 Gear manufacturing fundamentals (option 157)..... 393
Fundamentals..... 393
Please note while programming!..... 394

13.11 DEFINE GEAR (Cycle 285, DIN/ISO: G285, option 157)..... 395
Cycle run..... 395
Please note while programming!..... 395
Cycle parameters..... 396

13.12 GEAR HOBBING (Cycle 286, DIN/ISO: G286, option 157)..... 398
Application..... 398
Please note while programming!..... 399
Cycle parameters..... 400
Verifying and changing directions of rotation of the spindles..... 403

13.13 GEAR SKIVING (Cycle 287, DIN/ISO: G287, option 157)..... 404
Application..... 404
Please note while programming!..... 405
Cycle parameters..... 406
Verifying and changing directions of rotation of the spindles..... 408

13.14 MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)..... 409
Application..... 409
Please note while programming!..... 410
Cycle parameters..... 410

13.15 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143)..... 411
Cycle run..... 411
Please note while programming:..... 412
Cycle parameters..... 412

13.16 THREAD CUTTING (Cycle 18, DIN/ISO: G86)..... 413
Cycle run..... 413
Please note while programming:..... 414
Cycle parameters..... 415

13.17 Programming examples..... 416
Example: Interpolation turning with Cycle 291..... 416
Example: Interpolation Turning Cycle 292..... 419
Example of hob milling..... 421
Example of skiving..... 423

14 Cycles: Turning 425

14.1 Turning cycles (option 50) 426

Overview 426
Working with turning cycles 429
Blank form update (FUNCTION TURNDATA) 430

14.2 ADJUST XZ SYSTEM (Cycle 800, DIN/ISO: G800) 432

Application 432
Effect 435
Please note while programming: 436
Cycle parameters 438

14.3 RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801) 440

Please note while programming: 440
Effect 440
Cycle parameters 441

14.4 GEAR HOBBING (Cycle 880, DIN/ISO: G880, option 131) 442

Cycle run 442
Please note while programming: 444
Cycle parameters 446
Direction of rotation depending on the machining side (Q550) 449

14.5 CHECK UNBALANCE (Cycle 892, DIN/ ISO: G892) 450

Application 450
Please note while programming! 451
Cycle parameters 452

14.6 Fundamentals of Turning Cycles 453

14.7 TURN SHOULDER, LONGITUDINAL (Cycle 811, DIN/ISO: G812) 454

Application 454
Roughing cycle run 454
Finishing cycle run 454
Please note while programming: 455
Cycle parameters 456

14.8 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, ISO: G812) 457

Application 457
Roughing cycle run 457
Finishing cycle run 457
Please note while programming: 458
Cycle parameters 458

14.9 TURN PLUNGE CONTOUR LONGITUDINAL (Cycle 813, DIN/ISO: G813) 460

Application 460

Roughing cycle run ... 460
Finishing cycle run ... 460
Please note while programming: ... 461
Cycle parameters ... 462

14.10 TURN PLUNGE LONGITUDINAL EXTENDED (Cycle 814, DIN/ISO: G814) ... 463

Application ... 463
Roughing cycle run ... 463
Finishing cycle run ... 463
Please note while programming: ... 464
Cycle parameters ... 464

14.11 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G812) ... 466

Application ... 466
Roughing cycle run ... 466
Finishing cycle run ... 466
Please note while programming! ... 467
Cycle parameters ... 468

14.12 CONTOUR-PARALLEL TURNING (Cycle 815, DIN/ISO: G815) ... 470

Application ... 470
Roughing cycle run ... 470
Finishing cycle run ... 470
Please note while programming: ... 471
Cycle parameters ... 472

14.13 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821) ... 473

Application ... 473
Roughing cycle run ... 473
Finishing cycle run ... 473
Please note while programming: ... 474
Cycle parameters ... 475

14.14 TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822) ... 476

Application ... 476
Roughing cycle run ... 476
Finishing cycle run ... 476
Please note while programming: ... 477
Cycle parameters ... 478

14.15 TURN TRAVERSE PLUNGE (Cycle 823, DIN/ISO: G823) ... 480

Application ... 480
Roughing cycle run ... 480
Finishing cycle run ... 480
Please note while programming: ... 481
Cycle parameters ... 482

14.16 TURN PLUNGE TRANSVERSE EXTENDED (Cycle 824, DIN/ISO: G824)..... 483
Application..... 483
Roughing cycle run..... 483
Finishing cycle run..... 483
Please note while programming:..... 484
Cycle parameters..... 485

14.17 TURN CONTOUR TRANSVERSE (Cycle 820, DIN/ISO: G820)..... 487
Application..... 487
Roughing cycle run..... 487
Finishing cycle run..... 487
Please note while programming!..... 488
Cycle parameters..... 489

14.18 SIMPLE REC. TURNG., RADIAL DIR. (Cycle 841, DIN/ISO: G841)..... 491
Application..... 491
Roughing cycle run..... 491
Finishing cycle run..... 492
Please note while programming:..... 492
Cycle parameters..... 493

14.19 EXPANDED RECESS TURNING, RADIAL (Cycle 842, DIN/ISO: G842)..... 494
Application..... 494
Roughing cycle run..... 494
Finishing cycle run..... 495
Please note while programming:..... 495
Cycle parameters..... 496

14.20 SIMPLE RECESS TURNING, AXIAL (Cycle 851, DIN/ISO: G851)..... 498
Application..... 498
Roughing cycle run..... 498
Finishing cycle run..... 499
Please note while programming:..... 499
Cycle parameters..... 500

14.21 ENHANCED RECESS TURNING, AXIAL (Cycle 852, DIN/ISO: G852)..... 501
Application..... 501
Roughing cycle run..... 501
Finishing cycle run..... 502
Please note while programming:..... 502
Cycle parameters..... 503

14.22 CONTOUR RECESS TURNING, RADIAL (Cycle 840, DIN/ISO: G840)..... 505
Application..... 505
Roughing cycle run..... 505
Finishing cycle run..... 506

Please note while programming! ... 506
Cycle parameters ... 507

14.23 CONTOUR RECESS TURNING, AXIAL (Cycle 850, DIN/ISO: G850) ... 509

Application ... 509
Roughing cycle run ... 509
Finishing cycle run ... 510
Please note while programming: ... 510
Cycle parameters ... 511

14.24 SIMPLE RECESSING, RADIAL (Cycle 861, DIN/ISO: G861) ... 513

Application ... 513
Roughing cycle run ... 513
Finishing cycle run ... 513
Please note while programming: ... 514
Cycle parameters ... 514

14.25 EXPANDED RECESSING, RADIAL (Cycle 862, DIN/ISO: G862) ... 516

Application ... 516
Roughing cycle run ... 516
Finishing cycle run ... 516
Please note while programming: ... 517
Cycle parameters ... 518

14.26 SIMPLE RECESSING, AXIAL (Cycle 871, DIN/ISO: G871) ... 520

Application ... 520
Roughing cycle run ... 520
Finishing cycle run ... 520
Please note while programming: ... 521
Cycle parameters ... 521

14.27 EXPANDED RECESSING, AXIAL (Cycle 872, DIN/ISO: G872) ... 523

Application ... 523
Roughing cycle run ... 523
Finishing cycle run ... 524
Please note while programming: ... 524
Cycle parameters ... 525

14.28 CONTOUR RECESSING, RADIAL (Cycle 860, DIN/ISO: G860) ... 528

Application ... 528
Roughing cycle run ... 528
Finishing cycle run ... 528
Please note while programming! ... 529
Cycle parameters ... 530

14.29 CONTOUR RECESSING, AXIAL (Cycle 870, DIN/ISO: G870) ... 532

Application ... 532

Roughing cycle run ... 532
Finishing cycle run ... 532
Please note while programming! ... 533
Cycle parameters ... 534

14.30 THREAD, LONGITUDINAL (Cycle 831, DIN/ISO: G831) ... 536

Application ... 536
Cycle run ... 536
Please note while programming: ... 537
Cycle parameters ... 539

14.31 THREAD, EXTENDED (Cycle 832, DIN/ISO: G832) ... 540

Application ... 540
Cycle run ... 540
Please note while programming! ... 541
Cycle parameters ... 542

14.32 THREAD, CONTOUR-PARALLEL (Cycle 830, DIN/ISO: G830) ... 544

Application ... 544
Cycle run ... 544
Please note while programming: ... 545
Cycle parameters ... 546

14.33 TURNING, SIMULTANEOUS FINISHING (Cycle 883, DIN/ISO: G883, (option 158) ... 548

Application ... 548
Finishing cycle run ... 548
Please note while programming! ... 549
Cycle parameters ... 551

14.34 Programming example ... 553

Example: Gear hobbing ... 553
Example: Shoulder with recess ... 555
Example: turning, simultaneous finishing ... 558

15 Cycles: Grinding 561

15.1 Grinding cycles: general information 562

Overview 562
General information on the grinding cycles 563

15.2 DEFINE RECIP. STROKE (Cycle 1000, DIN/ISO: G1000, option 156) 565

Cycle run 565
Please note while programming! 566
Cycle parameters 567

15.3 START RECIP. STROKE (Cycle 1001, DIN/ISO: G1001, option 156) 568

Cycle run 568
Please note while programming! 568
Cycle parameters 568

15.4 STOP RECIP. STROKE (Cycle 1002, DIN/ISO: G1002, option 156) 569

Cycle run 569
Please note while programming! 569
Cycle parameters 569

15.5 DRESSING DIAMETER (Cycle 1010, DIN/ISO: G1010, option 156) 570

Cycle run 570
Please note while programming! 571
Cycle parameters 573

15.6 PROFILE DRESSING (Cycle 1015, DIN/ISO: G1015, option 156) 574

Cycle run 574
Please note while programming! 575
Cycle parameters 577

15.7 ACTIVATE WHEEL EDGE (Cycle 1030 DIN/ISO: G1030, option 156) 578

Cycle run 578
Please note while programming! 578
Cycle parameters 579

15.8 GRINDING WHL LENGTH COMPENSATION (Cycle 1032 DIN/ISO: G1032, option 156) 580

Cycle run 580
Please note while programming! 580
Cycle parameters 581

15.9 GRINDING WHL RADIUS COMPENSATION (Cycle 1033 DIN/ISO: G1033, option 156) 582

Cycle run 582
Please note while programming! 582
Cycle parameters 583

15.10 Programming examples 584
Example of grinding cycles 584
Example of dressing cycles 586
Example of a profile program 587

16 Using Touch Probe Cycles 589

16.1 General information about touch probe cycles 590

Method of function 590
Consideration of a basic rotation in the Manual operation mode 590
Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes 590
Touch probe cycles for automatic operation 591

16.2 Before You Start Working with Touch Probe Cycles 593

Maximum traverse to touch point: DIST in touch probe table 593
Set-up clearance to touch point: SET_UP in touch probe table 593
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table 593
Touch trigger probe, probing feed rate: F in touch probe table 594
Touch trigger probe, rapid traverse for positioning: FMAX 594
Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table 594
Executing touch probe cycles 595

16.3 Touch-probe table 596

General information 596
Editing touch probe tables 596
Touch probe data 597

17 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment 599

17.1 Overview 600

17.2 Touch probe cycles 14xx: fundamentals 601

Characteristics common to the 14xx touch probe cycles for measuring rotations 601
Semi-automatic mode 603
Evaluation of tolerances 607
Transferring the actual position 608

17.3 PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420) 609

Cycle run 609
Please note while programming! 610
Cycle parameters 611

17.4 PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410) 613

Cycle run 613
Please note while programming! 614
Cycle parameters 615

17.5 PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411) 617

Cycle run 617
Please note while programming! 618
Cycle parameters 619

17.6 Touch probe cycles 4xx: fundamentals 622

Characteristics common to all touch probe cycles for measuring workpiece misalignment 622

17.7 BASIC ROTATION (Cycle 400, DIN/ISO: G400) 623

Cycle run 623
Please note while programming: 623
Cycle parameters 624

17.8 ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401) 626

Cycle run 626
Please note while programming: 627
Cycle parameters 628

17.9 ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402) 630

Cycle run 630
Please note while programming: 631
Cycle parameters 632

17.10 ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403) 635

Cycle run 635
Please note while programming: 636
Cycle parameters 637

17.11 ROT. IN C AXIS (Cycle 405, DIN/ISO: G405) 640

Cycle run 640
Please note while programming: 641
Cycle parameters 642

17.12 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404) 644

Cycle run 644
Cycle parameters 644

17.13 Example: Determining a basic rotation from two holes 645

18 Touch Probe Cycles: Automatic Presetting 647

18.1 Fundamentals 648
Overview 648
Characteristics common to all touch probe cycles for presetting 650

18.2 PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410) 651
Cycle run 651
Please note while programming: 652
Cycle parameters 653

18.3 PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411) 655
Cycle run 655
Please note while programming: 656
Cycle parameters 657

18.4 PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412) 659
Cycle run 659
Please note while programming: 660
Cycle parameters 661

18.5 PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413) 664
Cycle run 664
Please note while programming: 665
Cycle parameters 666

18.6 PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414) 669
Cycle run 669
Please note while programming: 670
Cycle parameters 671

18.7 PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415) 674
Cycle run 674
Please note while programming: 675
Cycle parameters 676

18.8 PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416) 679
Cycle run 679
Please note while programming: 680
Cycle parameters 681

18.9 PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417) 683
Cycle run 683
Please note while programming: 683
Cycle parameters 684

18.10 PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418)..... 685
Cycle run..... 685
Please note while programming:..... 686
Cycle parameters..... 687

18.11 PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419)..... 690
Cycle run..... 690
Please note while programming:..... 690
Cycle parameters..... 691

18.12 SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408)..... 693
Cycle run..... 693
Please note while programming:..... 694
Cycle parameters..... 695

18.13 RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409)..... 697
Cycle run..... 697
Please note while programming:..... 698
Cycle parameters..... 699

18.14 Example: Presetting in center of a circular segment and on top surface of workpiece..... 701

18.15 Example: Presetting on top surface of workpiece and in center of a bolt hole circle.....702

19 Touch Probe Cycles: Automatic Workpiece Inspection 705

19.1 Fundamentals 706

Overview 706
Recording the results of measurement 707
Measurement results in Q parameters 709
Classification of results 709
Tolerance monitoring 709
Tool monitoring 710
Reference system for measurement results 711

19.2 REFERENCE PLANE (Cycle 0, DIN/ISO: G55) 712

Cycle run 712
Please note while programming: 712
Cycle parameters 712

19.3 POLAR PRESET (Cycle 1) 713

Cycle run 713
Please note while programming: 713
Cycle parameters 713

19.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420) 714

Cycle run 714
Please note while programming: 714
Cycle parameters 715

19.5 MEASURE HOLE (Circle 421, DIN/ISO: G421) 717

Cycle run 717
Please note while programming: 718
Cycle parameters 719

19.6 MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422) 722

Cycle run 722
Please note while programming: 723
Cycle parameters 724

19.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423) 727

Cycle run 727
Please note while programming: 728
Cycle parameters 729

19.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424) 731

Cycle run 731
Please note while programming: 731
Cycle parameters 732

19.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)..... 734
Cycle run..... 734
Please note while programming:..... 734
Cycle parameters..... 735

19.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)..... 737
Cycle run..... 737
Please note while programming:..... 737
Cycle parameters..... 738

19.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)..... 740
Cycle run..... 740
Please note while programming:..... 740
Cycle parameters..... 741

19.12 MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430)..... 743
Cycle run..... 743
Please note while programming:..... 744
Cycle parameters..... 744

19.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)..... 746
Cycle run..... 746
Please note while programming:..... 747
Cycle parameters..... 747

19.14 Programming Examples..... 749
Example: Measuring and reworking a rectangular stud..... 749
Example: Measuring a rectangular pocket and recording the results..... 751

20 Touch Probe Cycles: Special Functions 753

20.1 Fundamentals 754

Overview 754

20.2 MEASURING (Cycle 3) 755

Cycle run 755
Please note while programming: 755
Cycle parameters 756

20.3 MEASURING IN 3-D (Cycle 4) 757

Cycle run 757
Please note while programming: 757
Cycle parameters 758

20.4 PROBING IN 3-D (Cycle 444, ISO: G444) 759

Cycle run 759
Please note while programming! 761
Cycle parameters 762

20.5 FAST PROBING (Cycle 441, DIN/ISO: G441) 764

Cycle run 764
Please note while programming: 764
Cycle parameters 765

20.6 Calibrating a touch trigger probe 766

20.7 Displaying calibration values 767

20.8 TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461) 768

20.9 CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462) 770

20.10 TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463) 773

20.11 CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460) 776

21 VSC: camera-based setup control (option 136)..... 781

21.1 Camera-based Visual Setup Control VSC (Option 136)..... 782

Fundamentals..... 782
Managing monitoring data..... 784
Overview..... 786
Configuration..... 787
Defining the monitoring area..... 788
Result of the image evaluation..... 789

21.2 GLOBAL WORKING SPACE (Cycle 600, DIN/ISO: G600)..... 790

Application..... 790
Creating reference images..... 791
Monitoring phase..... 792
Please note while programming:..... 793
Cycle parameters..... 794

21.3 LOCAL WORKING SPACE (Cycle 601)..... 795

Application..... 795
Creating reference images..... 795
Monitoring phase..... 797
Please note while programming!..... 798
Cycle parameters..... 799

21.4 Possible queries..... 800

22 Touch Probe Cycles: Automatic Kinematics Measurement 801

22.1 Kinematics measurement with TS touch probes (option 48) 802

Fundamentals 802
Overview 803

22.2 Prerequisites 804

Please note while programming: 805

22.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48) 806

Cycle run 806
Please note while programming: 806
Cycle parameters 807
Log function 807
Notes on data management 808

22.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48) 809

Cycle run 809
Positioning direction 811
Machines with Hirth-coupled axes 812
Example calculation of measuring positions for an A axis: 812
Choice of number of measuring points 813
Choice of the calibration sphere position on the machine table 814
Notes on the accuracy 814
Notes on various calibration methods 815
Backlash 816
Please note while programming: 817
Cycle parameters 819
Various modes (Q406) 822
Log function 823

22.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48) 824

Cycle run 824
Please note while programming: 826
Cycle parameters 827
Adjustment of interchangeable heads 829
Drift compensation 831
Log function 833

22.6 KINEMATICS GRID (Cycle 453, DIN/ISO: G453, option 48) 834

Cycle run 834
Various modes (Q406) 836
Choice of the calibration sphere position on the machine table 836
Please note while programming: 837
Cycle parameters 838
Log function 840

23 Touch Probe Cycles: Automatic Tool Measurement 841

23.1 Fundamentals 842

Overview 842
Differences between Cycles 31 to 33 and Cycles 481 to 483 843
Setting machine parameters 844
Entries in the TOOL.T tool table 846

23.2 Calibrating the TT (Cycle 30 or 480, DIN/ISO: G480) 848

Cycle run 848
Please note while programming: 849
Cycle parameters 849

23.3 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481) 850

Cycle run 850
Please note while programming: 851
Cycle parameters 852

23.4 Measuring a tool radius (Cycle 32 or 482, ISO: G482) 854

Cycle run 854
Please note while programming: 855
Cycle parameters 856

23.5 Measuring tool length and radius (Cycle 33 or 483, ISO: G483) 858

Cycle run 858
Please note while programming: 859
Cycle parameters 860

23.6 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484) 862

Fundamentals 862
Cycle run 862
Please note while programming: 863
Cycle parameters 863

24 Tables of Cycles 865

24.1 Table of cycles 866

Fixed cycles 866
Turning cycles 868
Grinding cycles 869
Touch probe cycles 870

1

Fundamentals

1.1 About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine tool builder's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

⚠ DANGER

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

⚠ WARNING

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

⚠ CAUTION

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in things other than personal injury, such as property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard, e.g.: "There is danger of collision during subsequent machining operations"
- Escape – Hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.
In these instructions, you will find the following informational notes:

The information symbol indicates a **tip**.
A tip provides additional or supplementary information.

This symbol prompts you to follow the safety precautions of your machine tool builder. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine tool builder or other supplier.

Have you found any errors or would you like to suggest changes?

We are continuously striving to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

1.2 Control model, software, and features

This manual describes programming functions provided by controls as of the following NC software numbers.

Control model	NC software number
TNC 640	340590-10
TNC 640 E	340591-10
TNC 640 Programming station	340595-10

The suffix E indicates the export version of the control. The following software options are unavailable or only available to a limited extent in the export version:

- Advanced Function Set 2 (option 9) limited to four-axis interpolation
- KinematicsComp (option 52)

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

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

- Tool measurement with the TT

To find out about the actual features of your machine, please contact the machine manufacturer.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the HEIDENHAIN controls. Participation in one of these courses is recommended to familiarize yourself thoroughly with the control's functions.

Operating instructions:

All control functions not related to the cycles are described in the TNC 640 User's Manual. This manual is available from HEIDENHAIN upon request.

Conversational Programming User's Manual ID: 892903-xx

ISO Programming User's Manual ID: 892909-xx

User's Manual for Setup, Testing and Running NC programs ID: 1261174-xx

Software options

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

Additional Axis (options 0 to 7)	
Additional axis	Additional control loops 1 to 8
Advanced Function Set 1 (option 8)	
Expanded functions Group 1	**Machining with rotary tables** ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute **Coordinate conversions:** Tilting the working plane
Advanced Function Set 2 (option 9)	
Expanded functions Group 2 Export license required	**3-D machining:** ■ 3-D tool compensation through surface-normal vectors ■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = Tool Center Point Management) ■ Keeping the tool normal to the contour ■ Tool radius compensation normal to the tool direction ■ Manual traverse in the active tool-axis system **Interpolation:** Linear in > 4 axes (export license required)
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component
Dynamic Collision Monitoring – DCM (option 40)	
Dynamic Collision Monitoring	■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Collision monitoring in the Test Run mode ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements
CAD Import (option 42)	
CAD import	■ Support for DXF, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs

Adaptive Feed Control – AFC (option 45)	
Adaptive Feed Control	**Milling:** ■ 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 **Turning (option 50):** ■ Cutting force monitoring during machining
KinematicsOpt (option 48)	
Optimizing the machine kinematics	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics
Mill-Turning (option 50)	
Milling and turning modes	**Functions:** ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning cycles ■ Cycle 880: Gear hobbing (option 50 and option 131)
KinematicsComp (option 52)	
Three-dimensional compensation	Compensation of position and component errors
OPC UA NC Server 1 - 6 (Options 56 - 61)	
Standardized interface	The OPC UA NC Server provides a standardized interface (OPC UA) for external access to the control's data and functions These software options allow you to create up to six parallel client connections
3D-ToolComp (option 92)	
3-D tool radius compensation depending on the tool's contact angle Export license required	■ Compensate the deviation of the tool radius depending on the tool's contact angle ■ Compensation values in a separate compensation value table ■ Prerequisite: Working with surface normal vectors (**LN** blocks)
Extended Tool Management (option 93)	
Extended tool management	Python-based
Advanced Spindle Interpolation (option 96)	
Interpolating spindle	**Interpolation turning:** ■ Cycle 291: Interpolation turning, coupling ■ Cycle 292: Interpolation turning, contour finishing
Spindle Synchronism (option 131)	
Spindle synchronization	■ Synchronization of milling spindle and turning spindle ■ Cycle 880: Gear hobbing (option 50 and option 131)

Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface
Synchronizing Functions (option 135)	
Synchronization functions	**Real Time Coupling – RTC:** Coupling of axes
Visual Setup Control – VSC (option 136)	
Camera-based monitoring of the setup situation	■ Record the setup situation with a HEIDENHAIN camera system ■ Visual comparison of planned and actual status in the workspace
State Reporting Interface – SRI (option 137)	
HTTP accesses to the control status	■ Reading out the times of status changes ■ Reading out the active NC programs
Cross Talk Compensation – CTC (option 141)	
Compensation of axis couplings	■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of the TCP (**T**ool **C**enter **P**oint)
Position Adaptive Control – PAC (option 142)	
Adaptive position control	■ Changing of the control parameters depending on the position of the axes in the working space ■ Changing of the control parameters depending on the speed or acceleration of an axis
Load Adaptive Control – LAC (option 143)	
Adaptive load control	■ Automatic determination of workpiece weight and frictional forces ■ Changing of control parameters depending on the actual mass of the workpiece
Active Chatter Control – ACC (option 145)	
Active chatter control	Fully automatic function for chatter control during machining
Active Vibration Damping – AVD (option 46)	
Active vibration damping	Damping of machine oscillations to improve the workpiece surface
Batch Process Manager (option 154)	
Batch process manager	Planning of production orders
Component Monitoring (option 155)	
Component monitoring without external sensors	Monitoring configured machine components for overload

Grinding (Option 156)	
Jig grinding	■ Reciprocating stroke cycles ■ Cycles for dressing ■ Support of the "dressing tool" and "grinding tool" tool types
Gear Cutting (option 157)	
Machining gear systems	■ Cycle 285: Define gear wheel ■ Cycle 286: Gear hobbing ■ Cycle 287: Gear skiving
Advanced Function Set Turning (option 158)	
Advanced turning functions	Cycle 883: Simultaneous turning
Opt. contour milling (Option 167)	
Optimized contour cycles	■ Cycle 271: **OCM CONTOUR DATA** ■ Cycle 272: **OCM ROUGHING** ■ Cycle 273: **OCM FINISHING FLOOR** ■ Cycle 274: **OCM FINISHING SIDE**

Feature content level (upgrade functions)

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

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

Legal information

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

- Programming operating mode
- MOD function
- **License Info** softkey

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, they are always provided at the end of the cycle definition. For an overview of the optional Q parameters that have been added in this software version, refer to "New or changed cycle functions of software 34059x-10". You can decide for yourself whether you would like to define optional Q parameters or delete them with the NO ENT key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing NC programs after a software update, you can add optional Q parameters in cycles where needed. The following steps describe how this is done.

Proceed as follows:

- Call the cycle definition
- Press the right cursor key until the new Q parameters are displayed
- Confirm the displayed default value
- Alternatively, enter the desired value
- To load 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 load the new Q parameter, press the **NO ENT** key

Compatibility

Most NC programs created with older HEIDENHAIN contouring controls (as of TNC 150 B) can be run with the new software version of the TNC 640. Even if new optional parameters ("Optional parameters") have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way round, if you want to run an NC program created with a new software version 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 NC program is be downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

1.3 New or changed cycle functions of software 34059x-09

- New Cycle 285 DEFINE GEAR (option 157), see "DEFINE GEAR (Cycle 285, DIN/ISO: G285, option 157)", Page 395
- New Cycle 286 GEAR HOBBING (option 157), see "GEAR HOBBING (Cycle 286, DIN/ISO: G286, option 157)", Page 398
- New Cycle 287 GEAR SKIVING (option 157), see "GEAR SKIVING (Cycle 287, DIN/ISO: G287, option 157)", Page 404
- New Cycle 883 TURNING SIMULTANEOUS FINISHING (options 50 and 158), see "TURNING, SIMULTANEOUS FINISHING (Cycle 883, DIN/ISO: G883, (option 158) ", Page 548
- New Cycle 1410 PROBING ON EDGE (option 17), see "PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410)", Page 613
- New Cycle 1411 PROBING TWO CIRCLES (option 17),see "PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411)", Page 617
- New Cycle 1420 PROBING IN PLANE (option 17), see "PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420)", Page 609
- The simulation considers values sensed by a simulation probe. The simulation will be completed without error messages.
- In Cycle 24 SIDE FINISHING, a tangential helix will be used for rounding in the last infeed step, see "SIDE FINISHING (Cycle 24, DIN/ISO: G124)", Page 277
- Cycle 233 FACE MILLING, was extended by parameter Q367, SURFACE POSITION, see "FACE MILLING (Cycle 233, DIN/ISO: G233)", Page 206
- Cycle 257 CIRCULAR STUD, now uses Q207 FEED RATE MILLING for roughing, too, see "CIRCULAR STUD (Cycle 257, DIN/ISO: G257)", Page 196
- In Cycles 291 COUPLG.TURNG.INTERP. and 292 CONTOUR.TURNG.INTRP., the CfgGeoCycle configuration (no. 201000) is taken into account, see "COUPLING TURNING INTERPOLATION (Cycle 291, DIN/ISO: G291, option 96)", Page 364 see "INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ ISO : G292, option 96)", Page 371
- The touch probe cycles 408 to 419 consider chkTiltingAxes (no. 204600) for presetting, see "Touch Probe Cycles: Automatic Presetting", Page 647
- Touch probe cycles 41x, automatic presetting: new behavior of cycle parameters Q303 MEAS. VALUE TRANSFER and Q305 NUMBER IN TABLE, see "Touch Probe Cycles: Automatic Presetting", Page 647
- When pre-positioning the tool with Cycle 420 MEASURE ANGLE, the data both from the cycle and from the touch probe table will be taken into account, see "MEASURE ANGLE (Cycle 420, DIN/ISO: G420)", Page 714
- The help graphic in Cycle 444 PROBING IN 3-D for Q309 ERROR REACTION was changed; in addition, this Cycle now considers a TCPM, see "PROBING IN 3-D (Cycle 444, ISO: G444)", Page 759

- In Cycle 444 PROBING IN 3-D, the positions of the rotary axes relative to the tilt angles are checked depending on the setting of the optional machine parameter, see "PROBING IN 3-D (Cycle 444, ISO: G444)", Page 759
- Cycle 450 SAVE KINEMATICS, no longer writes identical values when restoring, see "SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48)", Page 806
- In Cycle 451 MEASURE KINEMATICS, the value 3 was added to cycle parameter Q406 MODE, see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 809
- In Cycles 451 MEASURE KINEMATICS and 453 KINEMATICS GRID, the radius of the calibration sphere is monitored in the second measurement only, see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 809 see "KINEMATICS GRID (Cycle 453, DIN/ISO: G453, option 48)", Page 834
- In Cycle 800 ADJUST XZ SYSTEM, the precision of the Q531 ANGLE OF INCIDENCE parameter was changed to 0.001°
- A REACTION column was added to the touch probe table, see "Touch-probe table", Page 596
- The CfgThreadSpindle machine parameter (no. 113600) is now available, see "TAPPING with a floating tap holder (Cycle 206, ISO: G206)", Page 129 , see "TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)", Page 132, see "TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209)", Page 136 , see "THREAD CUTTING (Cycle 18, DIN/ISO: G86)", Page 413

1.4 New or changed cycle functions of software 34059x-10

- New point pattern cycle 224 DATAMATRIX CODE PATTERN for the creation of a DataMatrix code, see "DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224) ", Page 251
- New cycle 238 MEASURE MACHINE STATUS for monitoring machine components for wear, see "MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)", Page 409
- New cycle 271 OCM CONTOUR DATA for defining machining information for the OCM cycles, see "OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167) ", Page 307
- New cycle 272 OCM ROUGHING for machining open pockets while maintaining the tool angle, see "OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167) ", Page 309
- New cycle 273 OCM FINISHING FLOOR for machining open pockets while maintaining the tool angle,see "OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167)", Page 312
- New cycle 274 OCM FINISHING SIDE for machining open pockets while maintaining the tool angle, see "OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167)", Page 314
- New cycles 1000 DEFINE RECIP. STROKE, 1001 START RECIP. STROKE, and 1002 STOP RECIP. STROKE for grinding with a reciprocation movement, see "DEFINE RECIP. STROKE (Cycle 1000, DIN/ISO: G1000, option 156)", Page 565, Page 568 and Page 569
- New cycles 1010 DRESSING DIAMETER and 1015 PROFILE DRESSING for dressing a grinding tool, see "DRESSING DIAMETER (Cycle 1010, DIN/ISO: G1010, option 156)", Page 570 and Page 574
- New cycle 1030 ACTIVATE WHEEL EDGE for activating the wheel edges, see "ACTIVATE WHEEL EDGE (Cycle 1030 DIN/ISO: G1030, option 156)", Page 578
- New cycles 1032 GRINDING WHL LENGTH COMPENSATION and 1033 GRINDING WHL RADIUS COMPENSATION for compensating the length and radius of a grinding tool, see "GRINDING WHL LENGTH COMPENSATION (Cycle 1032 DIN/ISO: G1032, option 156)", Page 580 and Page 582

- New DATUM TABLE soft key in the Program Run, Single Block and Program Run, Full Sequence operating modes. In addition, transfer of actual values to the datum table is possible in the Program Run, Single Block and Program Run, Full Sequence operating modes. see "Editing the datum table in the Program Run, Single Block or Program Run, Full Sequence operating mode", Page 224
- In cycles 205 UNIVERSAL PECKING and 241 SINGLE-LIP D.H.DRLNG, the input value for Q379 STARTING POINT is checked and compared to Q201 DEPTH. In case of conflict, an error message will be issued, see "UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)", Page 102 or Page 113
- Using cycle 225 ENGRAVING, it is possible to engrave the path or name of an NC program, see "Engraving the name and path of an NC program", Page 387
- If a limit has been programmed in cycle 233, the FACE MILLING cycle will extend the contour in the infeed direction by the corner radius, see "FACE MILLING (Cycle 233, DIN/ISO: G233)", Page 206
- Cycle 239 ASCERTAIN THE LOAD is only displayed if this has been defined by the machine tool builder, see "ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143)", Page 411
- The help graphics for Q224 ANGLE OF ROTATION in cycle 256 RECTANGULAR STUD was changed, see "RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)", Page 191
- The help graphics for Q326 SPACING IN 1ST AXIS and Q327 SPACING IN 2ND AXIS in cycle 415 PRESET INSIDE CORNER was changed, see "PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415)", Page 674
- Cycle 444 PROBING IN 3-D logs the measured 3-D distance. Thus, the control can distinguish between scrap and rework, see "PROBING IN 3-D (Cycle 444, ISO: G444)", Page 759
- The help graphics for Q341 PROBING THE TEETH in cycles 481 and 31 CAL. TOOL LENGTH and in cycles 482 and 32 CAL. TOOL RADIUS was changed, see "Measuring tool length (Cycle 31 or 481, DIN/ISO: G481)", Page 850 or Page 854
- In cycles 14xx, it is possible to use a handwheel for pre-positioning in semi-automatic mode. After probing, you can move to clearance height manually, see "Semi-automatic mode", Page 603

2

Fundamentals / Overviews

2.1 Introduction

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

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- You should run a 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. in **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 machining cycles with numbers 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 control assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.

If you want to delete a cycle that includes multiple sub-blocks, the control prompts you whether you want to delete the whole cycle.

2.2 Available cycle groups

Overview of fixed cycles

CYCL DEF

- Press the **CYCL DEF** key

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring and counterboring	84
DRILLING/ THREAD	Cycles for tapping, thread cutting and thread milling	128
POCKETS/ STUDS/ SLOTS	Cycles for milling pockets, studs and slots and for face milling	166
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	218
SL CYCLES	SL (Subcontour List) cycles for machining contours that consist of multiple overlapping subcontours as well as cycles for cylinder surface machining and trochoidal milling	260
PATTERN	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	244
TURNING	Cycles for turning and gear hobbing	426
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, determining the load, gear cycles	356
GRINDING	Cycles for grinding	562

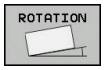
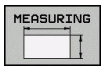
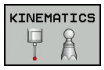
▷

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

Overview of touch probe cycles

TOUCH PROBE

- Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	599
	Cycles for automatic workpiece presetting	648
	Cycles for automatic workpiece inspection	706
	Special cycles	754
	Touch probe calibration	766
	Cycles for automatic kinematics measurement	803
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	842
	Cycles for VSC (camera-based setup control, option 136)	786

▷

- Switch to machine-specific touch probe cycles, if available; these touch probe cycles can be integrated by the machine tool builder

3

Using Fixed Cycles

3.1 Working with fixed cycles

Machine-specific cycles

Cycles are available for many machines. Your machine manufacturer can implement these cycles into the control, in addition to the HEIDENHAIN cycles. 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 functionality.

Some machine-specific cycles use transfer parameters that are also part of HEIDENHAIN standard cycles. In order to avoid problems (related to overwriting of transfer parameters that are used more than once), when using DEF-active cycles (cycles that the control runs automatically during cycle definition) and CALL-active cycles (cycles that you need to call to run them) used at the same time,

do the following:

Proceed as follows:

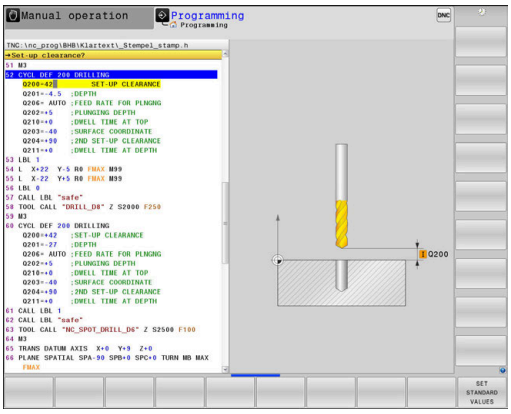
- Program DEF-active cycles before CALL-active cycles

Only program a DEF-active cycle between the definition of a CALL-active cycle and the cycle call if there are no interferences of transfer parameters of these two cycles.

Further information: "Calling a cycle", Page 64

Defining a cycle using soft keys

Proceed as follows:



- Press the **CYCL DEF** key

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

- Select the desired cycle group, e.g. drilling cycles

- Select the cycle, e.g. **THREAD MILLING**

> The control initiates a dialog and prompts you for all required input values. At the same time, a graphic is displayed in the right half of the screen. The required parameter is highlighted.

- Enter the required parameters
- Conclude each input with the **ENT** key

> The control closes the dialog when all required data has been entered.

Defining a cycle using the GOTO function

Proceed as follows:

- Press the **CYCL DEF** key

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

- Press the **GOTO** key

> The control opens the smartSelect selection window with an overview of the cycles.

- Select the desired cycle with the cursor keys or the mouse.
- Alternatively enter the cycle number
- Confirm each input with the **ENT** key

> The control then initiates the cycle dialog as described above.

Example

7 CYCL DEF 200 DRILLING	
Q200=2	**;SET-UP CLEARANCE**
Q201=3	**;DEPTH**
Q206=150	**;FEED RATE FOR PLNGNG**
Q202=5	**;PLUNGING DEPTH**
Q210=0	**;DWELL TIME AT TOP**
Q203=+0	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q211=0.25	**;DWELL TIME AT DEPTH**
Q395=0	**;DEPTH REFERENCE**

Calling a cycle

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Spindle direction of rotation (miscellaneous function **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 have been defined in the program. You cannot and must not call them:

- Cycle 220 for point patterns on circles and Cycle 221 for point patterns on lines
- SL Cycle 14 CONTOUR
- SL Cycle 20 CONTOUR DATA
- Cycle 32 TOLERANCE
- Cycles for coordinate transformation
- 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.

Proceed as follows:

CYCL CALL

- Press the **CYCL CALL** key
- Press the **CYCL CALL M** soft key
- If required, enter an M function (e.g. **M3**, to switch on the spindle)
- Press **END** to end the dialog

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 point table.

Further information: "Pattern definition with PATTERN DEF", Page 72

Further information: "Point tables", Page 79

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 control moves to the position defined in the **CYCL CALL POS** block:

- If the tool's current position in the tool axis is above the upper edge of the workpiece (**Q203**), the control first moves the tool to the programmed position in the working plane and then to the programmed position in the tool axis
- If the tool's current position in the tool axis is below the upper edge of the workpiece (**Q203**), the control first moves the tool to the clearance height in the tool axis and then to the programmed position in the working plane

Three coordinate axes must always be programmed in the **CYCL CALL POS** block. Using 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 is only used to traverse to the start position programmed in this block.

As a rule, the control 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 (e.g. 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 in the cycle as 0.

Calling a cycle with M89/M99

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

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

To cancel the effect of **M89**, proceed as follows:

- Program **M99** in the positioning block
- > The control moves to the last starting point.
- Alternatively, define a new machining cycle with **CYCL DEF**

The control does not support **M89** in combination with free programming of contours!

Calling a cycle with SEL CYCLE

With **SEL CYCLE**, you can call any NC program as a machining cycle.

Proceed as follows:

PGM CALL
- Press the **PGM CALL** key
- Press the **SEL CYCLE** soft key
- Press the **SELECT FILE** soft key
- Select an NC program

CYCL CALL
- Press the **CYCL CALL M**, **CYCL CALL PAT**, or **CYCL CALL POS** soft key
- Alternatively program **M99**

When you execute an NC program selected with **SEL CYCLE**, it will be executed in the Program Run, Single Block operating mode without stopping after each NC block. In addition, it is visible as a single NC block in the Program Run, Full Sequence operating mode.

Please note that **CYCL CALL PAT** and **CYCL CALL POS** use a positioning logic before executing the cycle. With respect to the positioning logic, **SEL CYCLE** and Cycle 12 **PGM CALL** show the same behavior: In point pattern cycles, the clearance height is calculated based on the maximum value of all Z positions existing at the start point of the pattern and all Z positions in the point pattern. With **CYCL CALL POS**, there will be no pre-positioning in the tool axis direction. This means that you need to manually program any pre-positioning in the file you call.

Working with a parallel axis

The control performs infeed movements in the parallel axis (W axis) that was defined in the **TOOL CALL** block as the spindle axis. The status display shows "W", the tool calculation is performed in the W axis.

This is only possible when programming the following cycles:

Cycle	W axis function
200 **DRILLING**	■
201 **REAMING**	■
202 **BORING**	■
203 **UNIVERSAL DRILLING**	■
204 **BACK BORING**	■
205 **UNIVERSAL PECKING**	■
208 **BORE MILLING**	■
225 **ENGRAVING**	■
232 **FACE MILLING**	■
233 **FACE MILLING**	■
241 **SINGLE-LIP D.H.DRLNG**	■

HEIDENHAIN recommends not to use **TOOL CALL W**! Use **FUNCTION PARAXMODE** or **FUNCTION PARAXCOMP**.

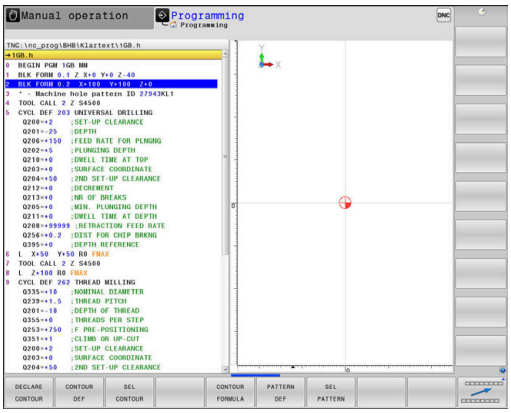
Further information: User's Manual for Conversational Programming

3.2 Program defaults for cycles

Overview

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

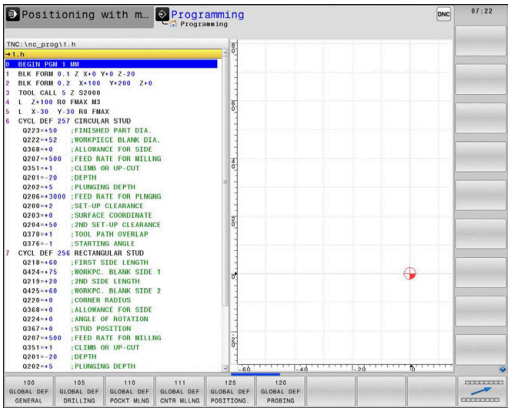
The following GLOBAL DEF functions are available:



Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	70
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	70
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	70
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	70
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	71
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	71

Entering GLOBAL DEF

Proceed as follows:



- Press the **PROGRAMMING** key

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM DEFAULTS

- Press the **PROGRAM DEFAULTS** soft key

GLOBAL DEF

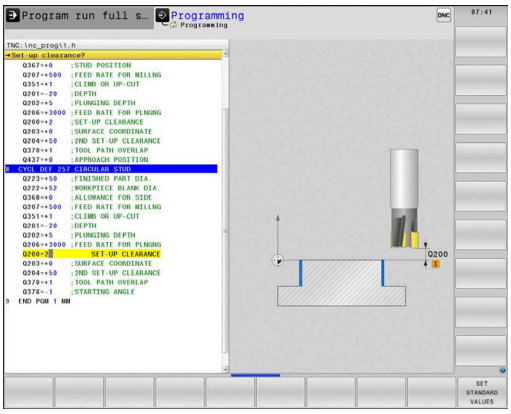
- Press the **GLOBAL DEF** soft key

100 GLOBAL DEF GENERAL

- Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
- Enter the required definitions
- Press the **ENT** key each time to confirm

Using GLOBAL DEF information

If you entered the respective GLOBAL DEF functions at the start of the program, you can reference these globally valid values when defining any machining cycle.



Proceed as follows:

- Press the **PROGRAMMING** key

- Press the **CYCL DEF** key

- Select the desired cycle group, e.g. drilling cycles

- Select the desired cycle, e.g. **DRILLING**.
> If a global parameter exists, the control will display the **SET STANDARD VALUES** soft key.

- Press the **SET STANDARD VALUES** soft key
> The control enters the word **PREDEF** in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program.

NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly.

- Make sure to use **GLOBAL DEF** carefully. Test your program before executing it
- If a fixed value is entered in machining cycles, **GLOBAL DEF** does not modify this value

Global data valid everywhere

- **SAFETY CLEARANCE**: Distance between tool face and workpiece surface for automated approach of the cycle start position in the tool axis
- **2ND SET-UP CLEARANCE**: Position to which the control moves the tool at the end of a machining step (the next machining position will be approached at this height in the working plane)
- **F POSITIONING**: Feed rate at which the control traverses the tool within a cycle
- **F RETRACTION**: Feed rate at which the control retracts the tool

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

Global data for drilling operations

- **RETRACT CHIP BREAKNG**: Value by which the control 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 stepover
- **CLIMB OR UP-CUT**: Climb or up-cut machining
- **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 stepover
- **CLIMB OR UP-CUT**: Climb or up-cut machining

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 a 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

- **SAFETY 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 control traverses the touch probe between measuring points if the **MOVE TO CLEARANCE** option is activated
- **MOVE TO CLEARANCE**: Select whether the control moves the touch probe to the set-up clearance or clearance height between the measuring points

The parameters apply to all touch probe cycles numbered 4xx.

3.3 Pattern definition with PATTERN DEF

Application

You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. Just like in cycle definitions, help graphics are available for pattern definition that clearly indicate the input parameters required.

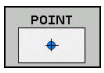
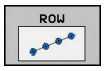
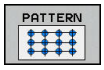
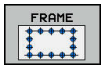
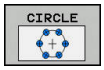
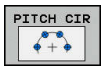
NOTICE

Danger of collision!

The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes For all tools axes apart from **Z** there is a danger of collision in the following operation!

- Use **PATTERN DEF** only in connection with the tool axis **Z**

The following machining patterns are available:

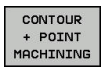
Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	74
	ROW Definition of a single row, straight or rotated	74
	PATTERN Definition of a single pattern, straight, rotated or distorted	75
	FRAME Definition of a single frame, straight, rotated or distorted	76
	CIRCLE Definition of a full circle	77
	PITCH CIRCLE Definition of a pitch circle	78

Entering PATTERN DEF

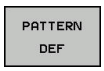
Proceed as follows:

- Press the **PROGRAMMING** key

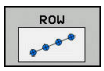
- Press the **SPEC FCT** key



- Press the **CONTOUR + POINT MACHINING** soft key



- Press the **PATTERN DEF** soft key



- Select the desired machining pattern, e.g. press the "single row" soft key
- Enter the required definitions
- Press the **ENT** key each time to confirm

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 64

The control 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.
Further information: User's Manual for Setup, Testing and Running NC programs

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

If the coordinate surface in PATTERN DEF is larger than in the cycle, the set-up clearance and the 2nd set-up clearance reference the coordinate surface in PATTERN DEF.

Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q352**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

Defining individual machining positions

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

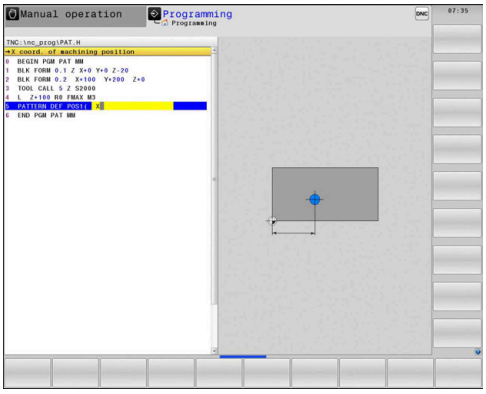
POS1 must be programmed with absolute coordinates. POS2 to POS9 can be programmed as absolute and/or incremental values.

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

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

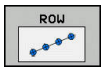
Example

```
10 L  Z+100 R0  FMAX
11 PATTERN DEF
    POS1 (X+25 Y+33.5  Z+0)
    POS2 (X+15 IY+6.5  Z+0)
```



Defining a single row

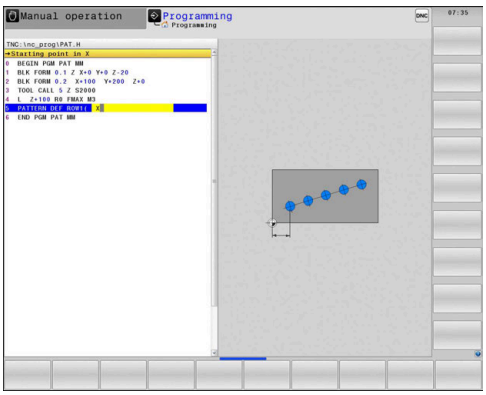
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



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

Example

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



Defining an individual 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.

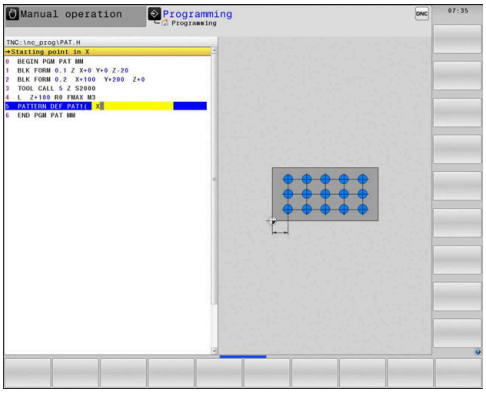
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.

PATTERN

- **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction. You can enter a positive or negative value
- **Spacing of machining positions Y** (incremental): Distance between the machining positions in the Y direction. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of rows**: Total number of rows in the pattern
- **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation about which only the principal axis of the working 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 about which only the secondary axis of the working 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

Example

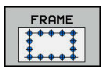
```
10 L  Z+100 R0  FMAX
11 PATTERN DEF  PAT1 (X+25 Y+33,5
     DX+8 DY+10  NUMX5 NUMY4 ROT+0
     ROTX+0 ROTY+0  Z+0)
```



Defining an individual frame

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.

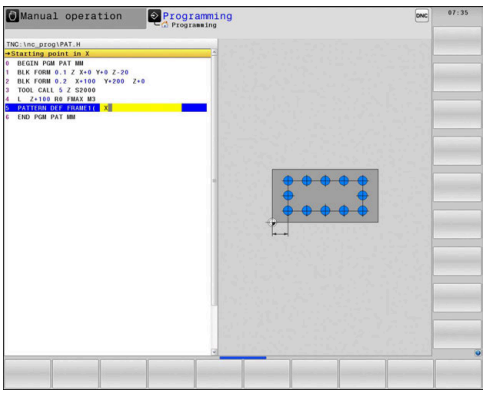
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



- **Starting point in X** (absolute): Coordinate of the frame starting point in the X axis
- **Starting point in Y** (absolute): Coordinate of the frame starting point in the Y axis
- **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction. You can enter a positive or negative value
- **Spacing of machining positions Y** (incremental): Distance between the machining positions in the Y direction. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of rows**: Total number of rows in the pattern
- **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation about which only the principal axis of the working 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 about which only the secondary axis of the working 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

Example

```
10 L  Z+100 R0  FMAX
11 PATTERN DEF  FRAME1
   (X+25  Y+33,5 DX+8  DY+10 NUMX5
   NUMY4 ROT+0  ROTX+0 ROTY+0  Z
   +0)
```



Defining a full circle

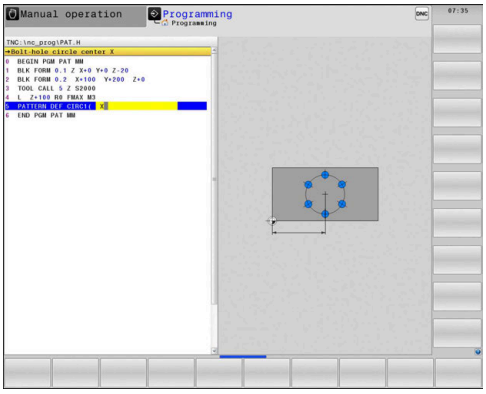
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

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 circular hole pattern
- **Starting angle**: Polar angle of the first machining position. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Number of operations**: Total number of machining positions on the circle
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

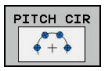
Example

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



Defining a pitch circle

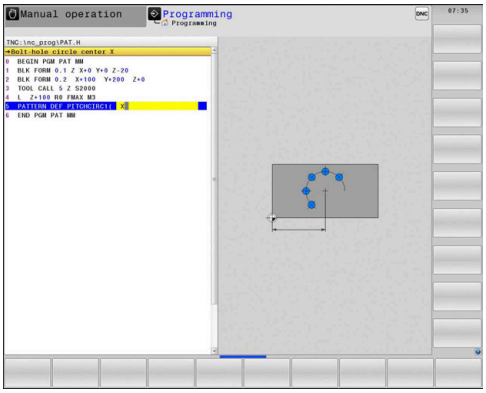
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



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

Example

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



3.4 Point tables

Application

You should create a point table whenever you want to run a cycle or several cycles in sequence, to machine 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 coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Entering values into a point table

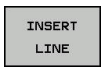
Proceed as follows:

- Press the **PROGRAMMING** key

- Press the **PGM MGT** key
- > The control opens the file manager.
- Select the folder in which you wish to create the new file
- Enter the name and file type (**.PNT**)

- Press the **ENT** key

- Press the **MM** or **INCH** soft key.
- > The control changes to the program window and displays an empty point table



- Press the **INSERT LINE** soft key to insert a new line
- Enter the coordinates of the desired machining position

Repeat the process until all desired coordinates have been entered.

If you intend to use the point table in SQL queries later, the table name must begin with a letter.

Use the **SORT/ HIDE COLUMNS** soft key to specify which coordinates you want to enter into 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.

Proceed as follows:

- Use the **CURSOR KEYS** to select the desired point in the table

- Select the **FADE** column

- To activate the hiding option for the point, press the **ENT** key

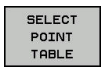
- To deactivate the hiding option for the point, press the **NO ENT** key

Selecting a point table in the NC program

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

Proceed as follows:

- Press the **PGM CALL** key



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

- Press the **SELECT FILE** soft key
- Select the point table
- Press the **OK** soft key

If the point table is not stored in the same directory as the NC program, you must enter the complete path.

Example

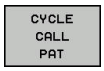
```
7 SEL PATTERN  "TNC:\DIRKT5\NUST35.PNT"
```

Calling a cycle in connection with point tables

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

Proceed as follows:

- Press the **CYCL CALL** key



- Press the **CYCL CALL PAT** soft key
- Enter a feed rate
- The control will use this feed rate to traverse between the points.
- Alternatively, press the **F MAX** soft key
- No input: the control will use the last programmed feed rate.
- Enter a miscellaneous function (M function) if required
- Confirm your input with the **END** key

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

Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q352**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

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

Effect of the point table with SL cycles and Cycle 12

The control interprets the points as an additional datum shift.

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

The control 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 as the starting point coordinate in the spindle axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

Effect of the point table with Cycles 251 to 254

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

NOTICE

Danger of collision!

If you program a clearance height for any points in a point table, the control will ignore the 2nd set-up clearance for **all** points of this machining cycle!

- Program GLOBAL DEF 125 POSITIONING beforehand. This will ensure that the control considers the clearance height from the point table for the corresponding point only.

If you call **CYCL CALL PAT**, the control will use the point table that you defined last. This is also the case if you defined the point table in an NC program nested with **CALL PGM**.

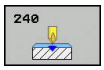
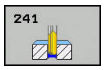
4

Fixed Cycles: Drilling

4.1 Fundamentals

Overview

The control provides the following cycles for all types of drilling operations:

Soft key	Cycle	Page
	240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth	121
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	85
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	87
	202 BORING With automatic pre-positioning, 2nd set-up clearance	89
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	92
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	98
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	102
	208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	110
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	113

4.2 DRILLING (Cycle 200, DIN/ISO: G200)

Cycle run

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

2 The tool drills to the first plunging depth at the programmed feed rate **F**

3 The control retracts the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth

4 The tool then drills deeper by the plunging depth at the programmed feed rate F.

5 The control repeats this procedure (steps 2 to 4) until the programmed depth is reached (the dwell time from **Q211** is effective with every infeed)

6 Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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

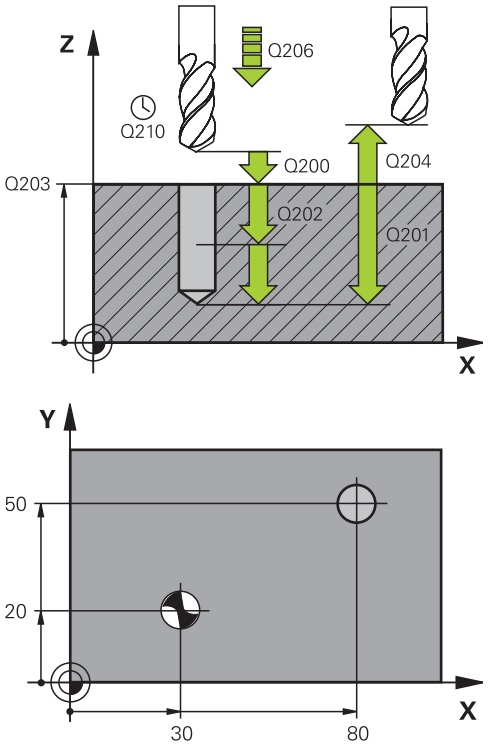
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range: –99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the control. Input range: 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth references the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool



Example

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

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

Cycle run

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

2 The tool reams to the entered depth at the programmed feed rate **F**.

3 If programmed, the tool remains at the hole bottom for the entered dwell time.

4 Then, the control retracts the tool at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

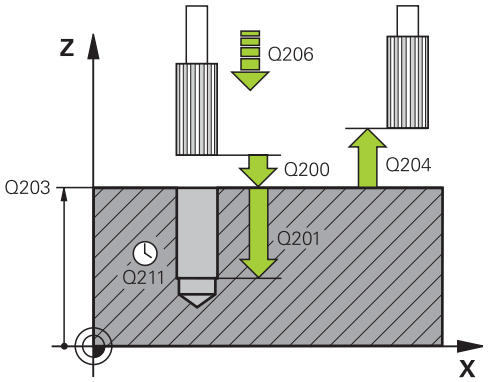
This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

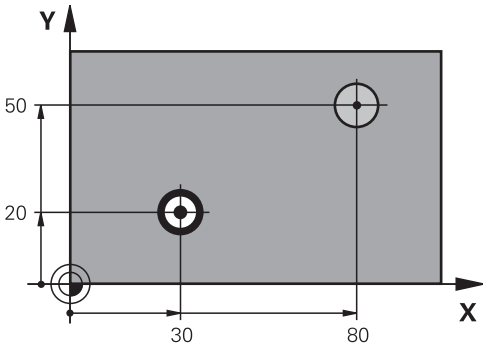
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

Cycle parameters

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





Example

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

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

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified 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 control then carries out an oriented spindle stop to the position that is defined in the **Q336** parameter
5 If retraction is selected, the control retracts in the programmed direction by 0.2 mm (fixed value)
6 The tool then retracts to set-up clearance at the retraction rate, and from there —if programmed—to the 2nd set-up clearance at **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**. If **Q214**=0 the tool tip remains on the wall of the hole
7 The control then returns the tool to the center of the hole

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

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

This cycle can only be used on machines with a servo-controlled spindle.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

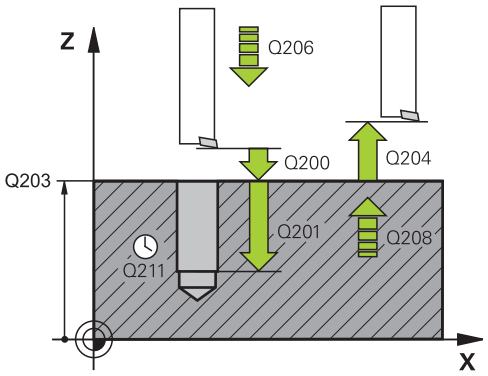
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

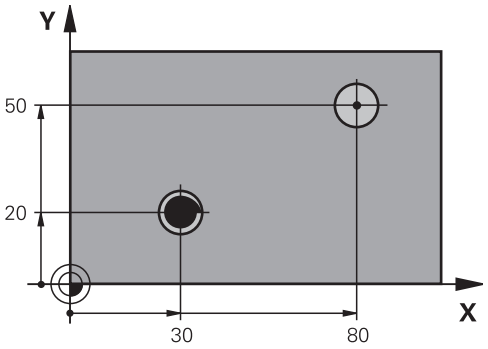
After machining, the control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool incrementally.

If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

Cycle parameters

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





Example

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

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

Cycle run

Behavior without chip breaking and without decrement:

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface

2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**

3 Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**

4 Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**

5 When machining without chip breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and remains there for the **DWELL TIME AT TOP Q210**

6 This sequence will be repeated until **depth Q201** is reached.

7 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and without decrement:

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface

2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**

3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**

4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**

5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**

6 If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**

7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached

8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached

9 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE** .The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and with decrement

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified **SAFETY CLEARANCE Q200** above the workpiece surface
2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** minus **DECREMENT Q212** at **FEED RATE FOR PLNGNG Q206**. The increasingly smaller difference between the updated **PLUNGING DEPTH Q202** minus **DECREMENT Q212** must never be smaller than **MIN. PLUNGING DEPTH Q205** (example: **Q202**=5, **Q212**=1, **Q213**=4, **Q205**= 3: The first plunging depth is 5 mm, the second plunging depth is 5 - 1 = 4 mm, the third plunging depth is 4 - 1 = 3 mm, the fourth plunging depth is also 3 mm)
5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6 If programmed, the control will now wait for the time specified in **DWELL TIME AT TOP Q210**
7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
9 If programmed, the control will now wait for the time specified in **DWELL TIME AT DEPTH Q211**
10 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE** .The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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

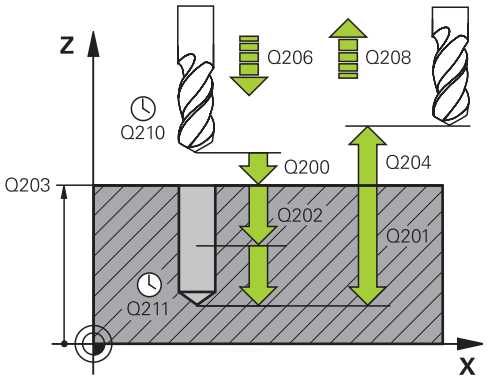
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range: –99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:

  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the control. Input range: 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999
- **Q213 Nr of breaks before retracting?**: Number of chip breaks before the control will retract the tool from the hole for chip removal. For chip breaking, the control retracts the tool each time by the value in **Q256**. Input range: 0 to 99999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999



Example

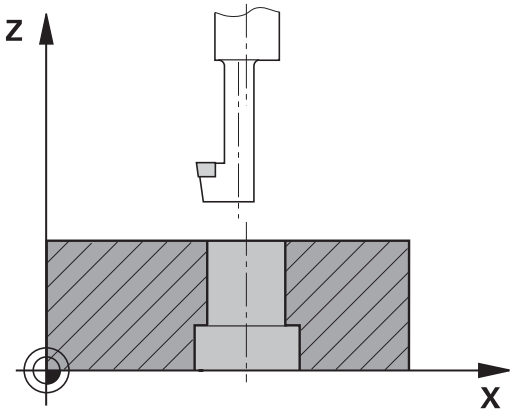
```
11 CYCL DEF 203  UNIVERSAL DRILLING
   Q200=2        ;SET-UP CLEARANCE
   Q201=-20      ;DEPTH
   Q206=150      ;FEED RATE FOR PLNGNG
   Q202=5        ;PLUNGING DEPTH
   Q211=0        ;DWELL TIME AT TOP
   Q203=+20      ;SURFACE COORDINATE
   Q204=50       ;2ND SET-UP CLEARANCE
   Q212=0.2      ;DECREMENT
   Q213=3        ;NR OF BREAKS
   Q205=3        ;MIN. PLUNGING DEPTH
   Q211=0.25     ;DWELL TIME AT DEPTH
   Q208=500      ;RETRACTION FEED RATE
   Q256=0.2      ;DIST FOR CHIP BRKNG
   Q395=0        ;DEPTH REFERENCE
12 L  X+30 Y+20  FMAX M3
13 CYCL CALL
```

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

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

Cycle run

This cycle allows counterbores to be machined from the underside of the workpiece.



1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface
2 The control 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 cutting edge has reached programmed set-up clearance beneath the lower workpiece edge
4 The control then centers the tool again in the bore hole, switches on the spindle and the coolant and moves at the feed rate for counterboring to the depth programmed for the counterbore
5 If programmed, the tool remains at the counterbore bottom. The tool will then be retracted from the hole again. The control carries out another oriented spindle stop and the tool is once again displaced by the off-center distance
6 Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**
7 The control then returns the tool to the center of the hole

Please note while programming:

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

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

This cycle can only be used on machines with a closed-loop spindle.

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

After machining, the control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool 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.

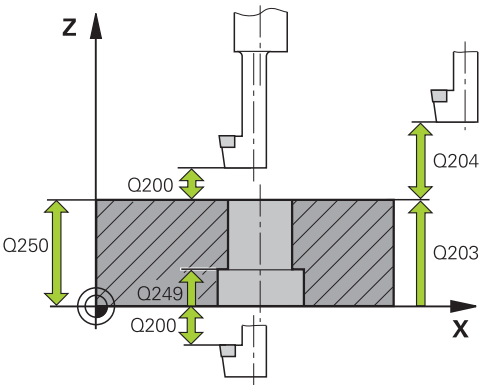
Enter the tool length so that the lower edge of the boring bar is measured, not the cutting edge.

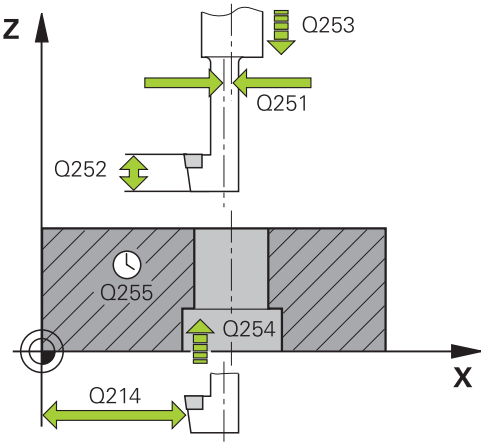
When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.

If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q249 Depth of counterbore?** (incremental): Distance between underside of workpiece and bottom 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
- **Q250 Material thickness?** (incremental): Thickness of the workpiece. Input range: 0.0001 to 99999.9999
- **Q251 Tool edge off-center distance?** (incremental): Off-center distance for the boring bar; value from the tool data sheet. Input range: 0.0001 to 99999.9999
- **Q252 Tool edge height?** (incremental): Distance between the underside of the boring bar and the main cutting edge; value from tool data sheet. Input range: 0.0001 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during countersinking. Input range: 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q255 Dwell time in secs.?**: Dwell time in seconds at the top of the bore hole. Input range: 0 to 3600.000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999





Example

11 CYCL DEF 204 BACK BORING	
Q200=2	;SET-UP CLEARANCE
Q249=+5	;DEPTH OF COUNTERBORE
Q250=20	;MATERIAL THICKNESS
Q251=3.5	;OFF-CENTER DISTANCE
Q252=15	;TOOL EDGE HEIGHT
Q253=750	;F PRE-POSITIONING

- **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the control displaces the tool by the off-center distance (after spindle orientation); programming 0 is not allowed
  1: Retract the tool in negative direction of the principal axis
  2: Retract the tool in negative direction of the secondary axis
  3: Retract the tool in positive direction of the principal axis
  4: Retract the tool in positive direction of the secondary axis
- **Q336 Angle for spindle orientation?** (absolute): Angle at which the control positions the tool before it is plunged into or retracted from the bore hole. Input range: –360.0000 to 360.0000

```
Q254=200  ;F COUNTERBORING
Q255=0    ;DWELL TIME
Q203=+20  ;SURFACE COORDINATE
Q204=50   ;2ND SET-UP CLEARANCE
Q214=1    ;DISENGAGING DIRECTN
Q336=0    ;ANGLE OF SPINDLE
```

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

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 If you enter a deepened starting point, the control moves at the defined positioning feed rate to the set-up clearance above the recessed starting point
3 The tool drills to the first plunging depth at the programmed feed rate **F**
4 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered advanced stop distance above the first plunging depth
5 The tool then drills deeper by the plunging depth at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
6 The control repeats this procedure (steps 2 to 4) until the 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 or the 2nd set-up clearance at the retraction feed rate. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

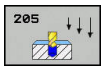
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

If you enter advance stop distances **Q258** not equal to **Q259**, the control will change the advance stop distances between the first and last plunging depths at the same rate.

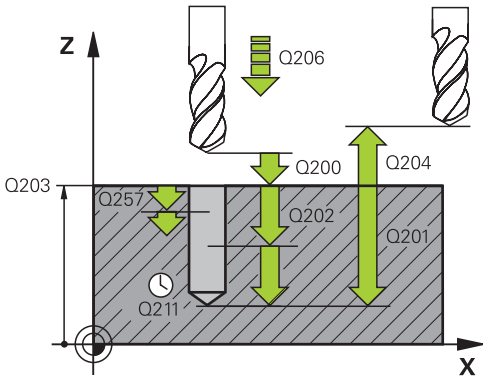
If you use **Q379** to enter a recessed starting point, the control will change the starting point of the infeed movement. Retraction movements are not changed by the control, they are always calculated with respect to the coordinate of the workpiece surface.

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range: –99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the control decreases the **Q202** plunging depth. Input range: 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999
- **Q258 Upper advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole. Input range: 0 to 99999.9999
- **Q259 Lower advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole; value for the last plunging depth. Input range: 0 to 99999.9999



Example

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

- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range: 0 to 99999.9999
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking. Input range: 0.000 to 99999.999
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000
- **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed starting point. Input range: 0 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input range: 0 to 99999.9999; alternatively **FMAX, FAUTO**
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth references the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point **Q379**. This value can be calculated as follows: 0.2 x **Q379**; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2

The starting point of drilling is calculated as follows: 0.2 x **Q379**=0.2*2=0.4; the starting point is 0.4 mm/inch above the recessed starting point. So if the recessed starting point is at –2, the control starts the drilling process at –1.6 mm.

The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (**Q200**=2, 5>2, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point **Q379**. This value can be calculated as follows: **0.8 x Q379**; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2

The position for chip removal is calculated as follows: 0.8 x **Q379**=0.8*2=1.6; the position for chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at –2, the control starts chip removal at –0.4.

The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (**Q200**=2, 8>2, so the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (**Q200**=2, 20>2, so the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (**Q200**=2, 80>2, so the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (**Q200**=5, 8>5, so the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (**Q200**=5, 20>5, so the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (**Q200**=5, 80>5, so the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (**Q200**=20, 80>20, so the value 20 is used.)	-80

4.8 BORE MILLING (Cycle 208, DIN/ISO: G208)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance **Q200** above the workpiece surface
2 In the next step, the control moves in a semi-circle to approach the helical path (starting at the center)
3 The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate **F**.
4 When the drilling depth is reached, the control once again traverses a full circle to remove the material remaining after the initial plunge.
5 The control then positions the tool at the center of the hole again, retracting it to the set-up clearance **Q200**.
6 This procedure is repeated until the nominal diameter is reached (the control calculates the stepover by itself)
7 Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance **Q204** at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

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

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

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

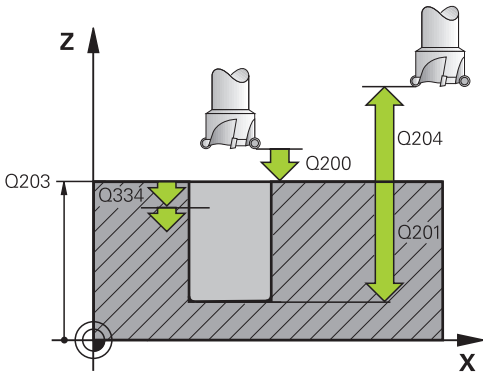
To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the **ANGLE** column of the tool table. The control then automatically calculates the max. infeed permitted and changes your entered value accordingly.

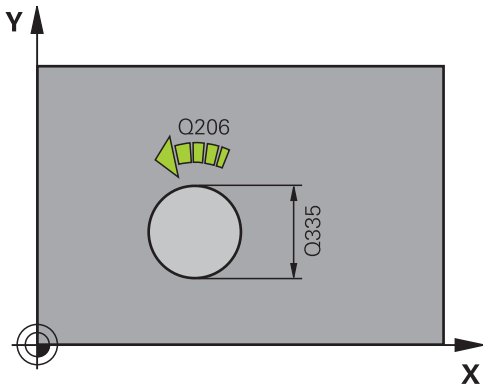
When calculating the infeed and the overlap factor, the control takes the corner radius DR2 of the current tool into account.

For the first helical path, the overlap factor is set as high as possible to prevent the tool from touching the ground of the hole. All other paths are distributed uniformly.

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range: –99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during helical drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q334 Feed per revolution of helix** (incremental): Depth of the tool plunge with each helix (=360°). Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q335 Nominal diameter?** (absolute): Hole diameter. If you entered the nominal diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation. Input range: 0 to 99999.9999
- **Q342 Roughing diameter?** (absolute): Enter the dimension of the predrilled diameter. Input range: 0 to 99999.9999
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)





Example

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

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

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **Safety clearance Q200** above the workpiece **SURFACE COORDINATE Q203**

2 Depending on the "Position behavior when working with Q379", Page 106, the control will either switch on the spindle with the programmed speed at the **Safety clearance Q200** or at a certain distance above the coordinate surface, see Page 106

3 The control executes the approach motion depending on the direction of rotation defined in the cycle with a spindle that rotates clockwise, counterclockwise, or is stationary

4 The tool drills to the hole depth at the feed rate **F**, or to the maximum 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 control reduces the feed rate by the feed rate factor after the dwell depth has been reached

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

6 The control repeats this procedure (steps 4 to 5) until the total hole depth is reached

7 After the control has reached this position, it will automatically switch off the coolant and set the speed to the value defined in **Q427 ROT.SPEED INFEED/OUT**

8 The control positions the tool to the retract position at the retraction feed rate. To find out the retract position value in your particular case, please refer to: see Page 106

9 If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

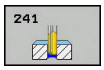
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

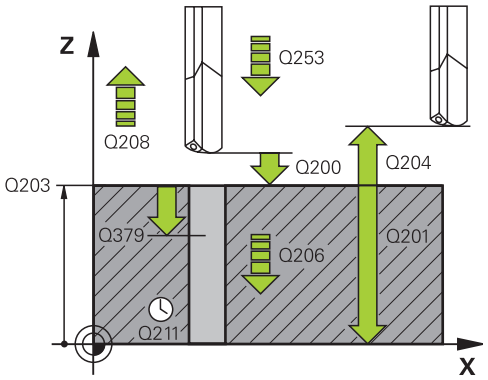
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and **Q203 SURFACE COORDINATE**. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between **Q203 SURFACE COORDINATE** and bottom of hole. Input range: –99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Distance to workpiece datum. Input range: –99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed starting point. Input range: 0 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the control retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- **Q426 Rot. dir. of entry/exit (3/4/5)?**: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input:
  3: Turn the spindle with M3
  4: Turn the spindle with M4
  5: Move with stationary spindle



Example

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

- **Q427 Spindle speed of entry/exit?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input range: 0 to 99999
- **Q428 Spindle speed for drilling?**: Desired speed for drilling. Input range: 0 to 99999
- **Q429 M function for coolant on?**: Miscellaneous function for switching on the coolant. The control switches the coolant on if the tool is in the hole at **Q379 STARTING POINT**. Input range: 0 to 999
- **Q430 M function for coolant off?**: Miscellaneous function for switching off the coolant. The control switches the coolant off if the tool is at **Q201 DEPTH**. Input range: 0 to 999
- **Q435 Dwell depth?** (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving the bottom of the hole in order to transport the chips to the top. Define a value smaller than **Q201 DEPTH**; input range: 0 to 99999.9999
- **Q401 Feed rate factor in %?**: Factor by which the control reduces the feed rate after **Q435 DWELL DEPTH** has been reached. Input range: 0 to 100
- **Q202 Maximum plunging depth?** (incremental): Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range: 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point **Q379**. This value can be calculated as follows: 0.2 x **Q379**; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2

The starting point of drilling is calculated as follows: 0.2 x **Q379**=0.2*2=0.4; the starting point is 0.4 mm/inch above the recessed starting point. So if the recessed starting point is at –2, the control starts the drilling process at –1.6 mm.

The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (**Q200**=2, 5>2, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point **Q379**. This value can be calculated as follows: **0.8 x Q379**; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2

The position for chip removal is calculated as follows: 0.8 x **Q379**=0.8*2=1.6; the position for chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at –2, the control starts chip removal at –0.4.

The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (**Q200**=2, 8>2, so the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (**Q200**=2, 20>2, so the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (**Q200**=2, 80>2, so the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (**Q200**=5, 8>5, so the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (**Q200**=5, 20>5, so the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (**Q200**=5, 80>5, so the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (**Q200**=20, 80>20, so the value 20 is used.)	-80

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

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified 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 is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

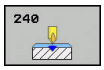
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

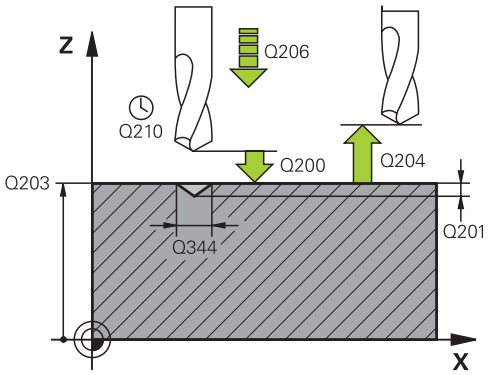
Program a positioning block for the starting point (hole center) in the working plane with the radius compensation **R0**

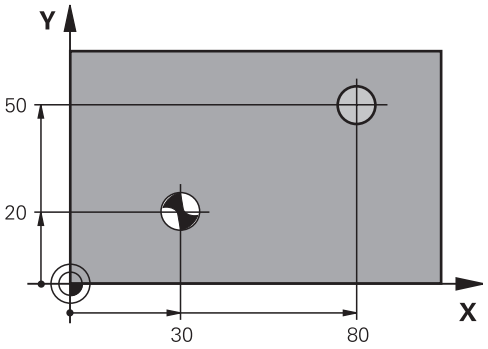
The algebraic sign for the **Q344** (diameter) or **Q201** (depth) cycle parameter determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

Cycle parameters



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



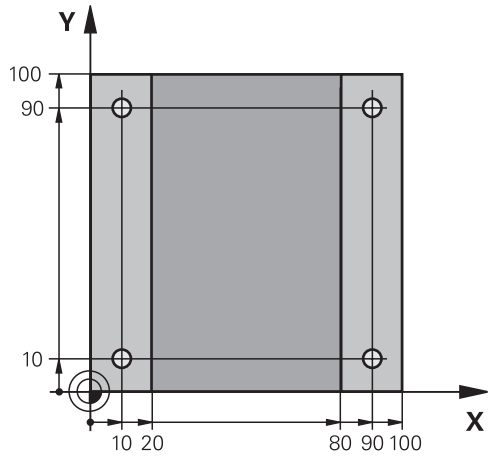


Example

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

4.11 Programming Examples

Example: Drilling cycles



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

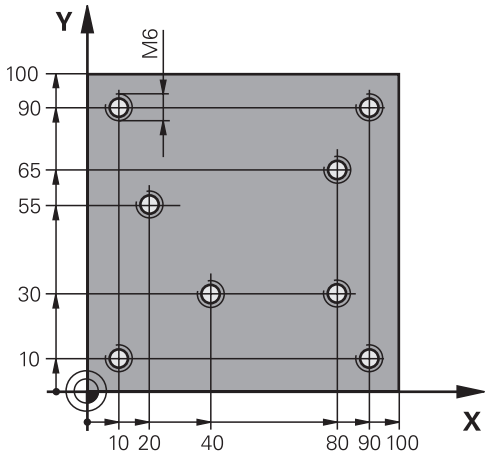
Example: Using drilling cycles in connection with PATTERN DEF

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

The tool radii have been selected in such a way 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)
  Further information: "Fundamentals", Page 128



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Tool call: centering tool (tool radius 4)
4 L Z+50 R0 FMAX	Move tool to clearance height
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 DIA./DEPTH	
Q201=-2 ;DEPTH	
Q344=-10 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
POSITION 7 GLOBAL DEF 125	This function is used for CYCL CALL PAT and positions the tool at the 2nd set-up clearance between the points. This function remains active until M30 is executed.
Q345=+1 ;SELECT POS. HEIGHT	
7 CYCL CALL PAT F5000 M13	Cycle call in connection with the point pattern

8 L Z+100 R0 FMAX	Retract the tool
9 TOOL CALL 2 Z S5000	Tool call: drill (radius 2.4)
10 L Z+50 R0 F5000	Move tool to clearance height
11 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
12 CYCL CALL PAT F500 M13	Cycle call in connection with the point pattern
13 L Z+100 R0 FMAX	Retract the tool
14 TOOL CALL Z S200	Tool call: tap (radius 3)
15 L Z+50 R0 FMAX	Move tool to clearance height
16 CYCL DEF 206 TAPPING	Cycle definition: tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
17 CYCL CALL PAT F5000 M13	Cycle call in connection with the point pattern
18 L Z+100 R0 FMAX M2	Retract the tool, end program
19 END PGM 1 MM	

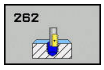
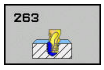
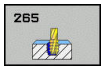
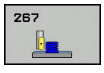
5

Fixed Cycles: Tapping / Thread Milling

5.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	129
	207 TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	132
	209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	136
	262 THREAD MILLING Cycle for milling a thread in pre-drilled material	143
	263 THREAD MILLING/ COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	147
	264 THREAD DRILLING/ MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool	151
	265 HELICAL THREAD DRILLING/MILLING Cycle for milling the thread into solid material	155
	267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer	159

5.2 TAPPING with a floating tap holder (Cycle 206, ISO: G206)

Cycle run

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

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

3 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

4 At the set-up clearance, the direction of spindle rotation reverses once again.

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

The algebraic sign for the DEPTH cycle parameter 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.

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

In Cycle 206, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

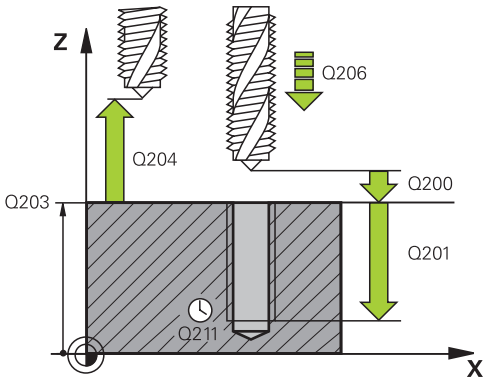
- **sourceOverride** (no. 113603):
  FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
  SpindlePotentiometer (feed rate override is not active) and
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters

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

Guide value: 4x pitch.

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



Example

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

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

F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the **NC Stop** key, the control will display a soft key with which you can retract the tool.

5.3 TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)

Cycle run

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

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool drills to the total hole depth in one movement.
3 It then reverses the direction of spindle rotation and the tool is retracted to the set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**
4 The control stops the spindle turning at that set-up clearance

Please note while programming!

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Machine and control 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

The spindle speed potentiometer is inactive.

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

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

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

For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

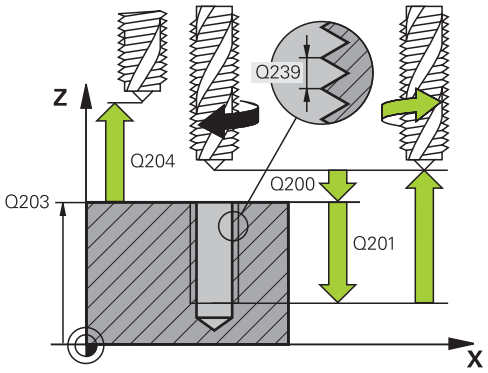
If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): SpindlePotentiometer (feed rate override is not active) and FeedPotentiometer (speed override is not active); the control then adjusts the spindle speed as required
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
- **limitSpindleSpeed** (no. 113604): Spindle speed limit
  True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time
  False: (Limiting not active)

Cycle parameters

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



Example

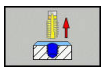
```
26 CYCL DEF 207  RIGID TAPPING NEU
   Q200=2        ;SET-UP CLEARANCE
   Q201=-20      ;DEPTH OF THREAD
   Q239=+1       ;THREAD PITCH
   Q203=+25      ;SURFACE COORDINATE
   Q204=50       ;2ND SET-UP CLEARANCE
```

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

- To interrupt thread cutting, press the **NC stop** key



- Press the retract soft key.

- Press **NC start**

> The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

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

Proceed as follows:

- To interrupt the program, press the **NC stop** key

- Press the **MANUAL TRAVERSE** soft key
- Retract the tool in the active spindle axis

- To continue program execution, press the **RESTORE POSITION** soft key

- Then press **NC start**

> The control returns the tool to the position it had assumed before the **NC stop** key was pressed.

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

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

Cycle run

The tool 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 or not.

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

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Machine and control 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

The spindle speed potentiometer is inactive.

If you defined a speed factor for fast retraction in cycle parameter **Q403**, the control limits the speed to the maximum speed of the active gear stage.

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

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

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

For tapping, the spindle and the tool axis are always synchronized with each other. Synchronization can take place while the spindle is stationary.

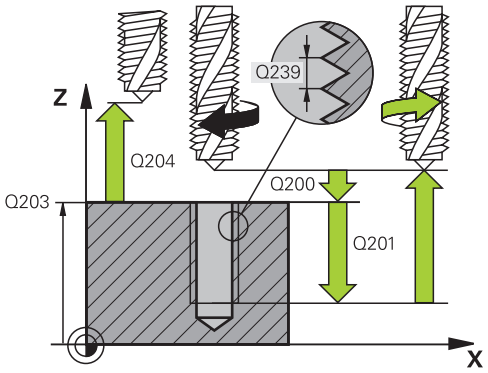
If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis can leave the acceleration path within this distance

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603):
  FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
  SpindlePotentiometer (feed rate override is not active) and
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range: -99999.9999 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range: 0 to 99999.9999
- **Q256 Retract dist. for chip breaking?**: The control multiplies the pitch **Q239** by the programmed value and retracts the tool by the calculated value for chip breaking. If you enter **Q256** = 0, the control retracts the tool completely from the hole (to the set-up clearance) for chip breaking. Input range: 0.000 to 99999.999
- **Q336 Angle for spindle orientation?** (absolute): Angle to which the control positions the tool before machining the thread. This allows you to recut the thread, if required. Input range: -360.0000 to 360.0000
- **Q403 RPM factor for retraction?**: Factor by which the control increases the spindle speed—and therefore also the retraction feed rate—when retracting from the hole. Input range: 0.0001 to 10. Maximum increase to maximum speed of the active gear stage.



Example

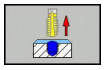
26 CYCL DEF 209 TAPPING W/ CHIP BRKG	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q239=+1	;THREAD PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=+1	;DIST FOR CHIP BRKNG
Q336=50	;ANGLE OF SPINDLE
Q403=1.5	;RPM FACTOR

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

- To interrupt thread cutting, press the **NC stop** key



- Press the retract soft key.

- Press **NC start**

> The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

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

Proceed as follows:

- To interrupt the program, press the **NC stop** key

- Press the **MANUAL TRAVERSE** soft key
- Retract the tool in the active spindle axis

- To continue program execution, press the **RESTORE POSITION** soft key

- Then press **NC start**

> The control returns the tool to the position it had assumed before the **NC stop** key was pressed.

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

5.5 Fundamentals of Thread Milling

Prerequisites

- Your machine tool features 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 can set the compensation in **TOOL CALL** using the **DR** delta radius).
- Cycles 262, 263, 264, and 267 can only be used with clockwise rotating tools, cycle 265 is suitable for clockwise or counterclockwise rotating tools
- The working direction is determined by the following input parameters: Algebraic sign **Q239** (+ = right-hand thread / – = left-hand thread) and type of milling **Q351** (+1 = climb / –1 = up-cut).

  The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

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

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

NOTICE

Danger of collision!

If you program the plunging depth values with different algebraic signs a collision may occur.

- Make sure to program all depth values with the same algebraic sign. Example: If you program the **Q356** COUNTERSINKING DEPTH parameter with a negative sign, then **Q201** DEPTH OF THREAD must also have a negative sign
- If you would like to repeat just the counterbore procedure in a cycle, you can enter 0 for DEPTH OF THREAD. In this case, the machining direction is determined by the programmed COUNTERSINKING DEPTH

NOTICE

Danger of collision!

A collision may occur if, upon tool breakage, you retract the tool from the hole in the direction of the tool axis only.

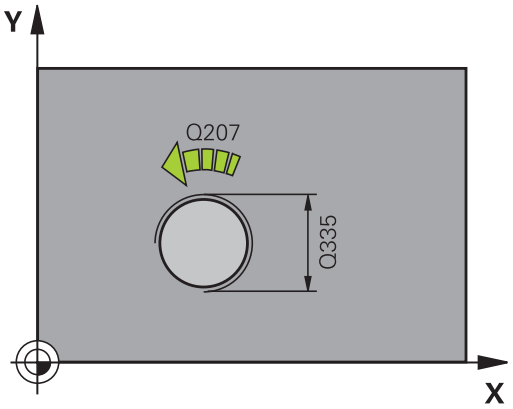
- Stop the program run if the tool breaks
- Change to Positioning with Manual Data Input mode of operation
- First move the tool in a linear movement towards the hole center
- Retract the tool in the tool axis direction

The programmed feed rate for thread milling references the cutting edge of the tool. However, since the control always displays the feed rate relative to the center 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.

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

Cycle run



1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3 The tool then approaches the nominal thread diameter tangentially in a helical movement. Before the helical approach, a compensating movement 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 control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

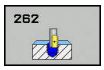
If you program the thread depth =0, the cycle will not be executed.

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 tool diameter is smaller than the nominal thread diameter by four times the thread pitch.

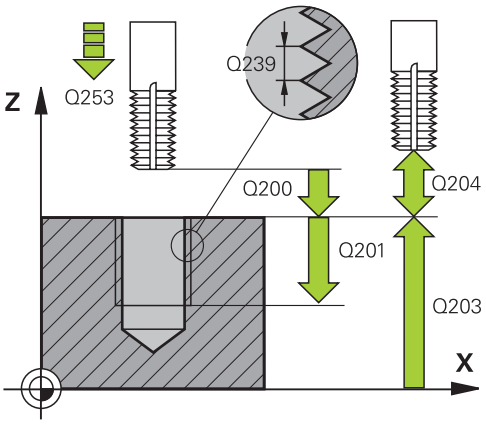
Note that the control makes a compensating movement in the tool axis before the approach. The length of the compensation movement is at most half of the thread pitch. Ensure sufficient space in the hole!

If you change the thread depth, the control will automatically move the starting point for the helical movement.

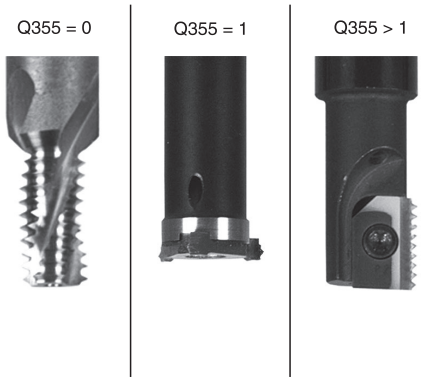
Cycle parameters



- **Q335 Nominal diameter?**: Nominal thread diameter. Input range: 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range: -99999.9999 to 99999.9999
- **Q355 Number of threads per step?**: Number of turns by which the tool is displaced:
  0 = one helix over the entire thread depth
  1 = continuous helix over the entire thread length
  >1 = several helix paths with approach and departure, between these, the control offsets the tool by **Q355** x pitch. Input range: 0 to 99999



- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**
- **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range: 0 to 99999.999 alternatively **FAUTO**



Example

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

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

Cycle run

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

Countersinking

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

3 If a set-up clearance to the side has been entered, the control immediately positions the tool at the pre-positioning feed rate to the countersinking depth.

4 Then, depending on the available space, the control smoothly approaches the tool to the core diameter, either tangentially from the center or with a pre-positioning movement 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 control positions the tool without compensation from its center position 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 control 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 algebraic sign of 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 control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

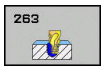
The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Countersinking depth
3. Depth at front

If you program one of the depth parameter to be 0, the control does not execute that step.

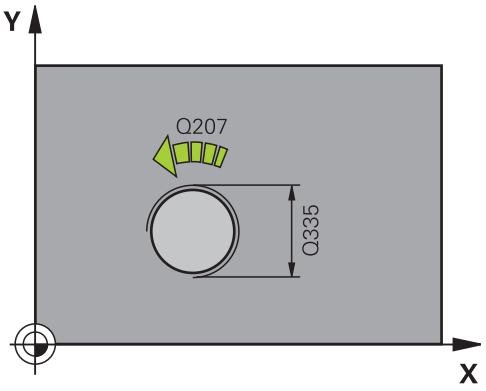
If you want to countersink at front, define the countersinking depth as 0.

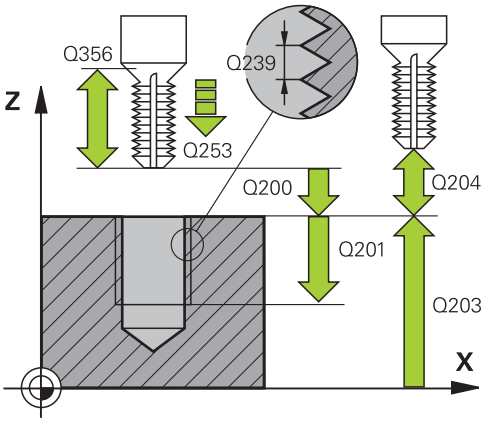
Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

Cycle parameters

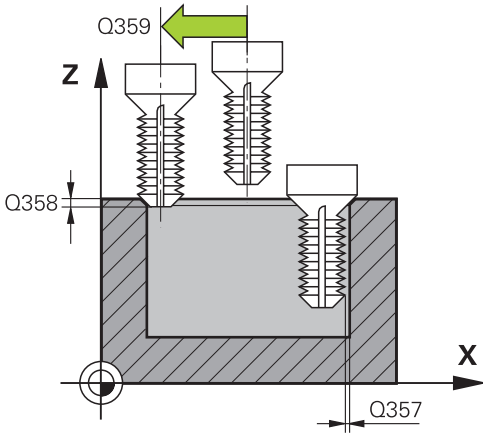


- **Q335 Nominal diameter?**: Nominal thread diameter. Input range: 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of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





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



Example

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

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

Cycle run

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

Drilling

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

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

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

5 The control repeats this procedure (steps 2 to 4) until the 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 control positions the tool without compensation from its center position 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 control 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 algebraic sign of 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 control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

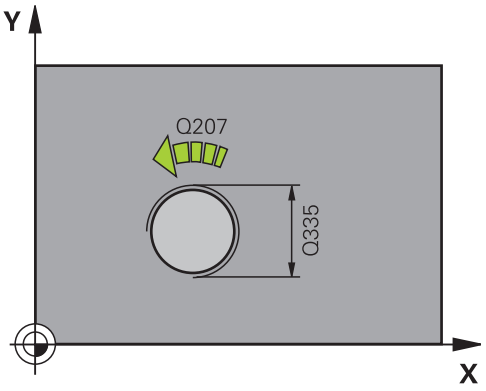
If you program one of the depth parameter to be 0, the control does not execute that step.

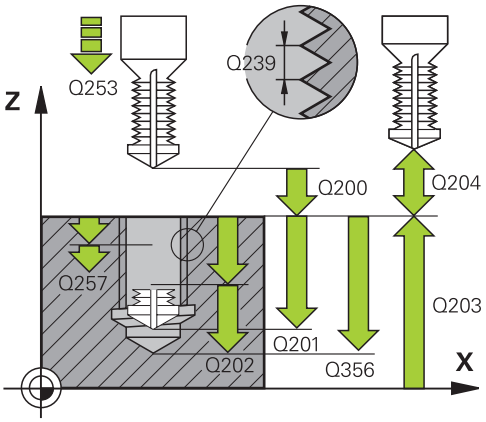
Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

Cycle parameters

- **Q335 Nominal diameter?**: Nominal thread diameter. Input range: 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range: -99999.9999 to 99999.9999
- **Q356 Total hole depth?** (incremental): Distance between workpiece surface and hole bottom. Input range: -99999.9999 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)
- **Q202 Maximum plunging depth?** (incremental): Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range: 0 to 99999.9999

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q258 Upper advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole. Input range: 0 to 99999.9999





Example

```
25 CYCL DEF 264  THREAD DRILLNG/
   MLLNG
   Q335=10     ;NOMINAL DIAMETER
   Q239=+1.5   ;THREAD PITCH
   Q201=-16    ;DEPTH OF THREAD
   Q356=-20    ;TOTAL HOLE DEPTH
   Q253=750    ;F PRE-POSITIONING
   Q351=+1     ;CLIMB OR UP-CUT
```

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

```
Q202=5      ;PLUNGING DEPTH
Q258=0.2    ;UPPER ADV STOP DIST
Q257=5      ;DEPTH FOR CHIP BRKNG
Q256=0.2    ;DIST FOR CHIP BRKNG
Q358=+0     ;DEPTH AT FRONT
Q359=+0     ;OFFSET AT FRONT
Q200=2      ;SET-UP CLEARANCE
Q203=+30    ;SURFACE COORDINATE
Q204=50     ;2ND SET-UP CLEARANCE
Q206=150    ;FEED RATE FOR PLNGNG
Q207=500    ;FEED RATE MILLING
Q512=0      ;FEED FOR APPROACH
```

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

Cycle run

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

Countersinking at front

2 If countersinking occurs before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking occurs after thread milling, the control moves the tool to the countersinking depth at the feed rate for prepositioning

3 The control positions the tool without compensation from its center position 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 control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread

6 The tool then approaches the nominal thread diameter tangentially in a helical movement

7 The tool moves on a continuous helical downward path until the thread depth value is reached

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 control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

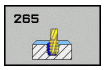
The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Depth at front

If you program one of the depth parameter to be 0, the control does not execute that step.

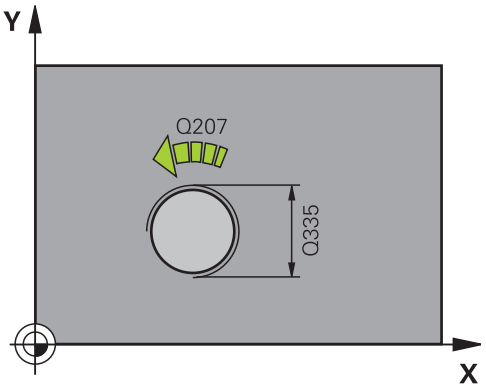
If you change the thread depth, the control will automatically move the starting point for the helical movement.

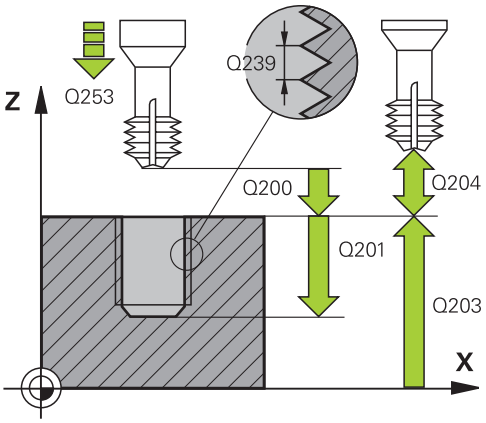
The type of milling (up-cut or climb) is determined by the thread (right-hand or left-hand thread) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

Cycle parameters

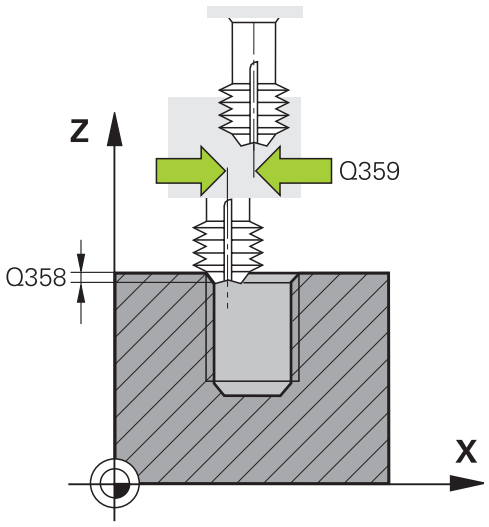


- **Q335 Nominal diameter?**: Nominal thread diameter. Input range: 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of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





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



Example

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

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

Cycle run

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

Countersinking at front

2 The control approaches the starting point for countersinking at front, starting from the center of the stud, on the reference axis in the working plane. 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 control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking

5 The tool then moves on a semicircle to the starting point

Thread milling

6 The control positions the tool at 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 nominal 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 control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

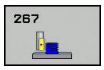
The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

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

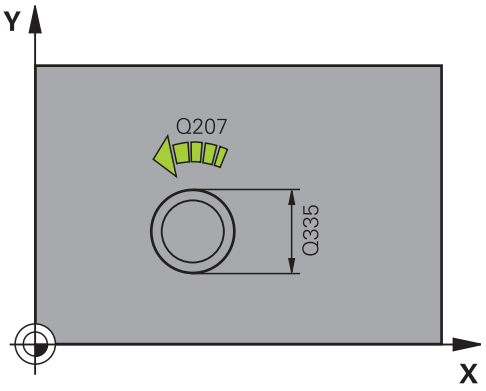
If you program one of the depth parameter to be 0, the control does not execute that step.

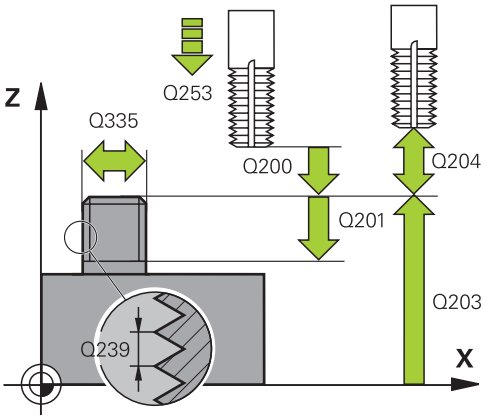
The algebraic sign for the cycle parameter "thread depth" determines the working direction.

Cycle parameters

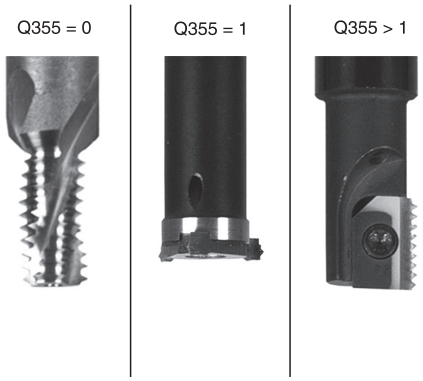


- **Q335 Nominal diameter?**: Nominal thread diameter. Input range: 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range: -99999.9999 to 99999.9999
- **Q355 Number of threads per step?**: Number of turns by which the tool is displaced:
  0 = one helix over the entire thread depth
  1 = continuous helix over the entire thread length
  >1 = several helix paths with approach and departure, between these, the control offsets the tool by **Q355** x pitch. Input range: 0 to 99999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999





- **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range: -99999.9999 to 99999.9999
- **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during countersinking. Input range: 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**
- **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range: 0 to 99999.999 alternatively **FAUTO**



Example

25 CYCL DEF 267 OUTSIDE THREAD MLLNG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-20	;DEPTH OF THREAD
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 MILLING
Q512=0	;FEED FOR APPROACH

5.11 Programming Examples

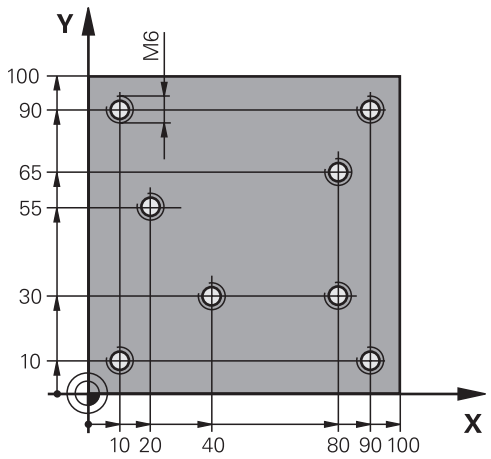
Example: Thread milling

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

The tool radii have been selected in such a way that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call: centering tool
4 L Z+10 R0 F5000	Move tool to clearance height (program a value for F): the control positions the tool at the clearance height after every cycle
5 SEL PATTERN "TAB1"	Select the point table
6 CYCL DEF 240 CENTERING	Cycle definition: centering
Q200=2 ;SET-UP CLEARANCE	
Q343=1 ;SELECT DIA./DEPTH	
Q201=-3.5 ;DEPTH	
Q344=-7 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q11=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT; feed rate between the points: 5000 mm/min
11 L Z+100 R0 FMAX M6	Retract the tool
12 TOOL CALL 2 Z S5000	Tool call: drill
13 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	

Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
15 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
16 L Z+100 R0 FMAX M6	Retract the tool
17 TOOL CALL 3 Z S200	Tool call: tap
18 L Z+50 R0 FMAX	Move tool to clearance height
19 CYCL DEF 206 TAPPING	Cycle definition: 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	

TAB1. PNT point table

TAB1. PNTMM
NRXYZ
0 +10 +10 +0
1 +40 +30 +0
2 +90 +10 +0
3 +80 +30 +0
4 +80 +65 +0
5 +90 +90 +0
6 +10 +90 +0
7 +20 +55 +0
[END]

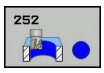
6

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

6.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	167
	252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	173
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal plunging	180
	254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal plunging	185
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	191
	257 CIRCULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	196
	258 POLYGON STUD Roughing/finishing cycle for machining a regular polygon	200
	233 FACE MILLING Machining the face with up to three limits	206

6.2 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 control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowance (**Q368** and **Q369**) into account.

3 At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall, then moves to the set-up clearance above the current pecking depth and returns from there at rapid traverse to the pocket center.

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

Finishing

5 If finishing allowances have been defined, the control plunges and then approaches the contour. The approach movement occurs on a radius in order to ensure a gentle approach. The control first finishes the pocket walls, with multiple infeeds, if so specified.

6 Then the control finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially

Please note while programming!

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.

Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

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

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

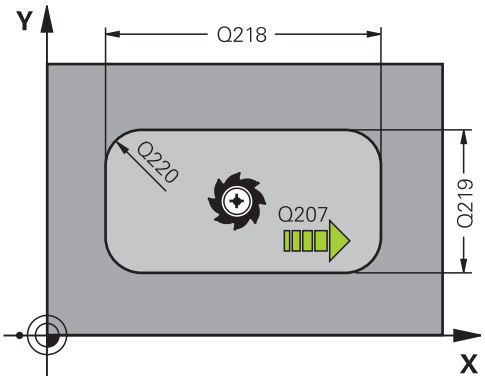
Program a sufficient set-up clearance so that the tool cannot jam because of chips.

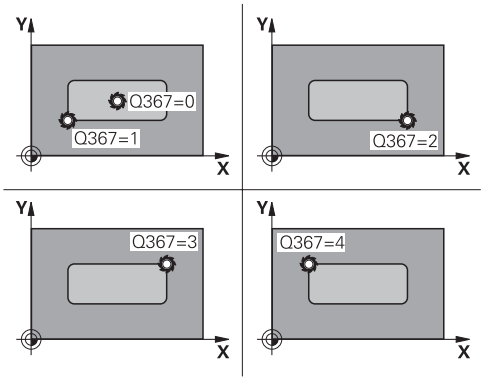
The control outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter (no. 201006).

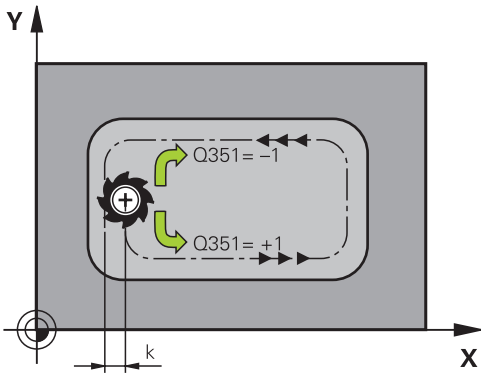
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

Cycle parameters

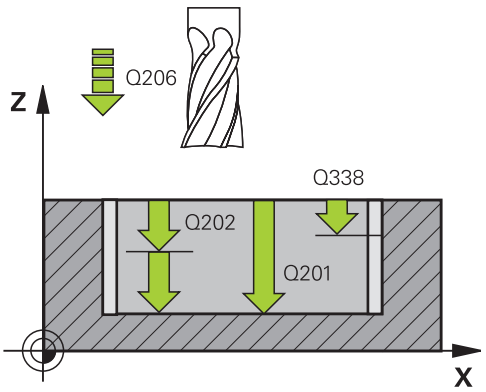
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q218 First side length?** (incremental): Pocket length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q219 Second side length?** (incremental): Pocket length, parallel to the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q220 Corner radius?**: Radius of the pocket corner. If you have entered 0 here, the control assumes that the corner radius is equal to the tool radius. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q224 Angle of rotation?** (absolute): Angle by which the entire machining configuration is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: –360.0000 to 360.0000
- **Q367 Position of pocket (0/1/2/3/4)?**: Position of the pocket with respect to the position of the tool when the cycle is called:
  0: Tool position = Pocket center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range: -99999.9999 to 99999.9999

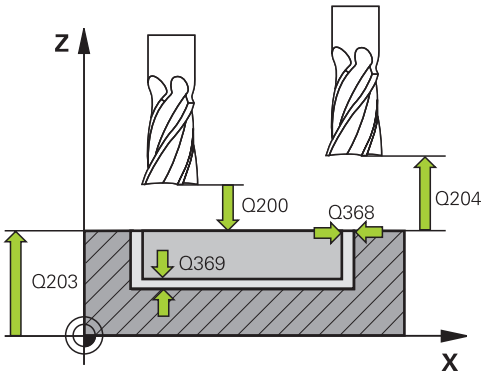






- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. Input range: 0.0001 to 1.9999; alternatively **PREDEF**
- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  0: vertical plunging. The control plunges the tool 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 control generates an error message
  2: reciprocating plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. The reciprocation length depends on the plunging angle. As a minimum value, the control uses twice the tool diameter
  PREFEF: The control uses the value from the GLOBAL DEF block





Example

```
8 CYCL DEF 251  RECTANGULAR POCKET
   Q215=0       ;MACHINING OPERATION
   Q218=80      ;FIRST SIDE LENGTH
   Q219=60      ;2ND SIDE LENGTH
   Q220=5       ;CORNER RADIUS
   Q368=0.2     ;ALLOWANCE FOR SIDE
   Q224=+0      ;ANGLE OF ROTATION
   Q367=0       ;POCKET POSITION
   Q207=500     ;FEED RATE MILLING
   Q351=+1      ;CLIMB OR UP-CUT
   Q201=-20     ;DEPTH
   Q202=5       ;PLUNGING DEPTH
   Q369=0.1     ;ALLOWANCE FOR FLOOR
   Q206=150     ;FEED RATE FOR PLNGNG
   Q338=5       ;INFEED FOR FINISHING
   Q200=2       ;SET-UP CLEARANCE
   Q203=+0      ;SURFACE COORDINATE
   Q204=50      ;2ND SET-UP CLEARANCE
   Q370=1       ;TOOL PATH OVERLAP
   Q366=1       ;PLUNGE
   Q385=500     ;FINISHING FEED RATE
   Q439=0       ;FEED RATE REFERENCE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

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

6.3 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 control 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 control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowance (**Q368** and **Q369**) into account.
4 At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall to 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 was 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 second set-up clearance **Q204** in the tool axis and returns at rapid traverse to the pocket center.

Finishing

1 If finishing allowances have been defined, the control first finishes the pocket walls, in multiple infeeds, if so specified.
2 The control positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200**
3 The control roughs out the pocket from the inside out, until the diameter **Q223** is reached
4 Then, the control again positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200** and repeats the finishing procedure for the side wall at the new depth
5 The control repeats this process until the programmed diameter is reached
6 After machining to the diameter **Q223**, the control retracts the tool tangentially by the finishing allowance **Q368** plus the set-up clearance **Q200** in the working plane, then retracts it at rapid traverse to the set-up clearance **Q200** in the tool axis and returns it to the pocket center.
7 Next, the control moves the tool in the tool axis to the depth **Q201** and finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially.
8 The control 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:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, 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 control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

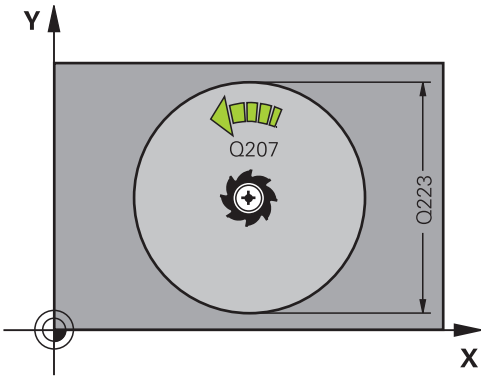
Program a sufficient set-up clearance so that the tool cannot jam because of chips.

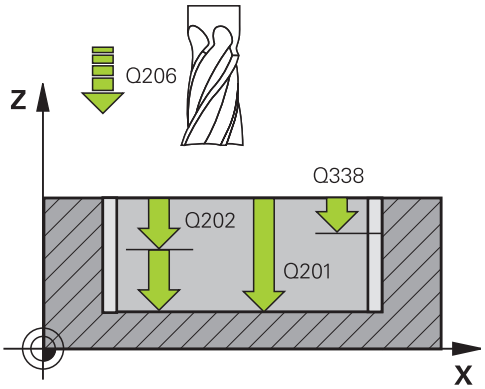
The control outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter (no. 201006).

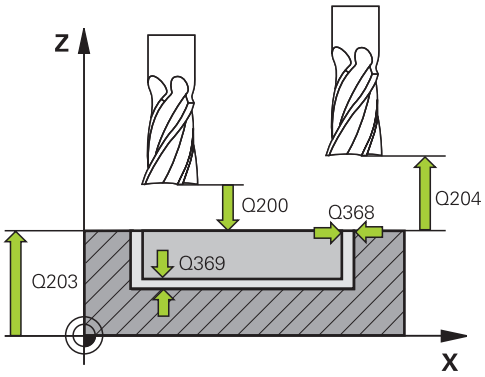
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

Cycle parameters

- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q223 Circle diameter?**: Diameter of the finished pocket. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range: -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**







- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.1 to 1.9999; alternatively **PREDEF**
- **Q366 Plunging strategy (0/1)?**: Type of plunging strategy:
  - 0 = vertical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as 0 or 90. Otherwise, the control will 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. Otherwise, the control will display an error message
  - Alternative: **PREDEF**

Example

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

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

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

Cycle run

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

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

Roughing

1 Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs out the slot from the inside out, taking the finishing allowances (**Q368** and **Q369**) into account
3 The control retracts the tool to the set-up clearance **Q200**. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4 This process is repeated until the programmed slot depth is reached.

Finishing

5 If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially in the left slot arc
6 Then the control finishes the floor of the slot from the inside out.

Please note while programming:

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

- Do **not** program any incremental dimensions after this cycle
- Program an absolute position in all main axes after this cycle

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, 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 control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

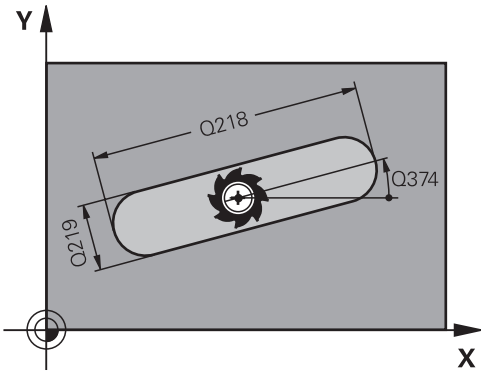
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

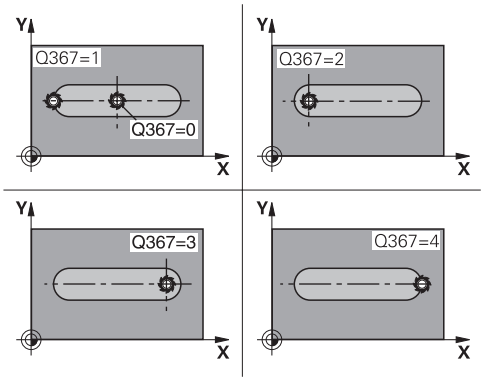
If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

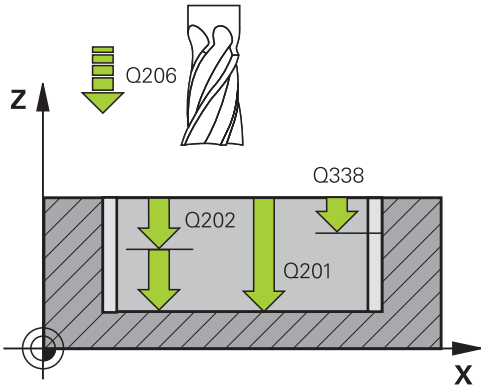
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

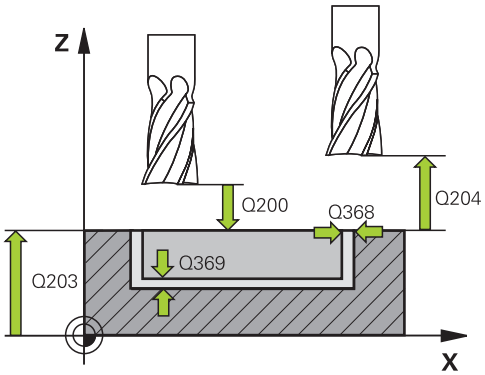
Cycle parameters

- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q218 Length of slot?** (value parallel to the principal axis of the working plane): Enter the length of the slot. Input range: 0 to 99999.9999
- **Q219 Width of slot?** (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q374 Angle of rotation?** (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: –360.000 to 360.000
- **Q367 Position of slot (0/1/2/3/4)?**: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position = slot center
  1: Tool position = left end of slot
  2: Tool position = center of left slot arc
  3: Tool position = center of right slot arc
  4: Tool position = right end of slot
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: –99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999









- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999

Example

```
8 CYCL DEF 253  SLOT MILLING
    Q215=0      ;MACHINING OPERATION
    Q218=80     ;SLOT LENGTH
    Q219=12     ;SLOT WIDTH
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q374=+0     ;ANGLE OF ROTATION
    Q367=0      ;SLOT POSITION
    Q207=500    ;FEED RATE MILLING
    Q351=+1     ;CLIMB OR UP-CUT
```

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

```
Q201=-20    ;DEPTH
Q202=5      ;PLUNGING DEPTH
Q369=0.1    ;ALLOWANCE FOR FLOOR
Q206=150    ;FEED RATE FOR PLNGNG
Q338=5      ;INFEED FOR FINISHING
Q200=2      ;SET-UP CLEARANCE
Q203=+0     ;SURFACE COORDINATE
Q204=50     ;2ND SET-UP CLEARANCE
Q366=1      ;PLUNGE
Q385=500    ;FINISHING FEED RATE
Q439=0      ;FEED RATE REFERENCE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

6.5 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 control roughs out the slot from the inside out, taking the finishing allowances (**Q368** and **Q369**) into account
3 The control retracts the tool to the set-up clearance **Q200**. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4 This process is repeated until the programmed slot depth is reached.

Finishing

5 If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially
6 Then the control finishes the floor of the slot from the inside out

Please note while programming:

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

- Do not program any incremental dimensions after this cycle
- Program an absolute position in all main axes after this cycle

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, 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 control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

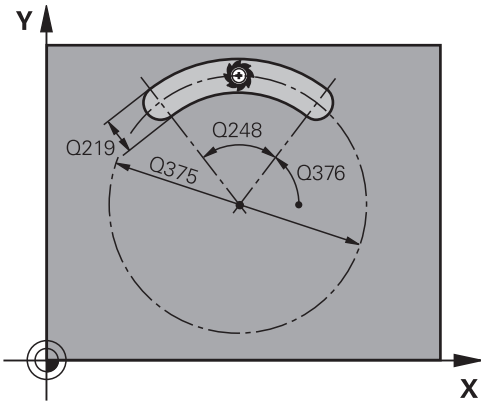
If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

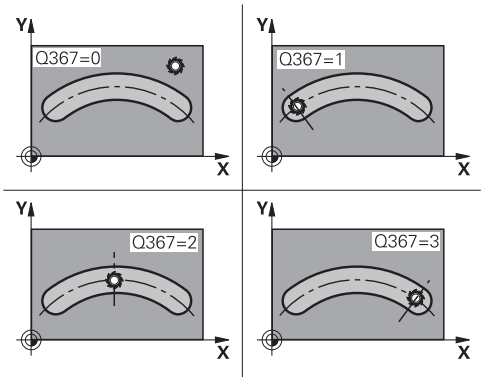
The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

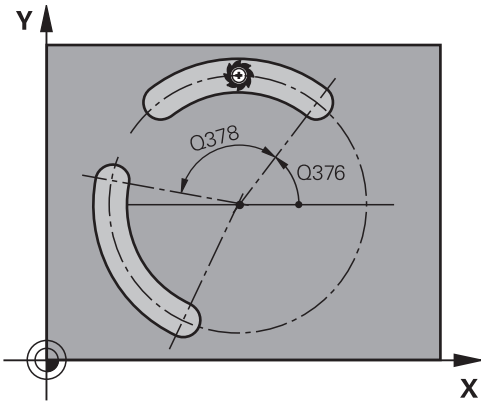
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

Cycle parameters

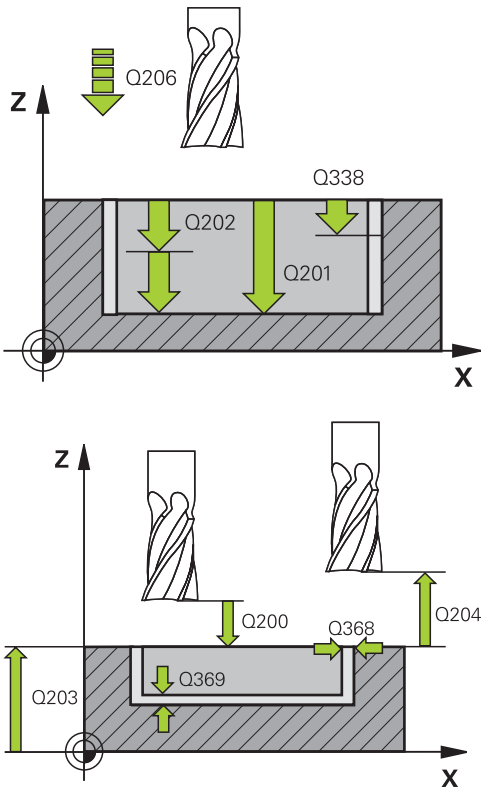
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q219 Width of slot?** (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q375 Pitch circle diameter?**: Enter the diameter of the pitch circle. Input range: 0 to 99999.9999
- **Q367 Ref. for slot pos. (0/1/2/3)?**: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle
  1: Tool position = center of left slot arc. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account
  2: Tool position = center of centerline. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account
  3: Tool position = center of right slot arc. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account.
- **Q216 Center in 1st axis?** (absolute): Center of the pitch circle in the principal axis of the working plane. **Only effective if Q367 = 0**. Input range: –99999.9999 to 99999.9999







- **Q217 Center in 2nd axis?** (absolute): Center of the pitch circle in the secondary axis of the working plane. **Only effective if Q367 = 0**. Input range: –99999.9999 to 99999.9999
- **Q376 Starting angle?** (absolute): Enter the polar angle of the starting point. Input range: –360.000 to 360.000
- **Q248 Angular length?** (incremental): Enter the angle between the starting point and the end point of the slot. Input range: 0 to 360.000
- **Q378 Intermediate stepping angle?** (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. Input range: –360.000 to 360.000
- **Q377 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range: 1 to 99999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: –99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**



Example

8 CYCL DEF 254 CIRCULAR SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q375=80	;PITCH CIRCLE DIAMETR
Q367=0	;REF. SLOT POSITION
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q376=+45	;STARTING ANGLE
Q248=90	;ANGULAR LENGTH
Q378=0	;STEPPING ANGLE
Q377=1	;NR OF REPETITIONS

- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  0: vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
  1, 2: reciprocal plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control generates an error message
  PREDEF: The control uses the value from the GLOBAL DEF block
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
  0: Feed rate with respect to the tool center point path
  1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
  2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
  3: Feed rate always refers to the cutting edge

```
Q207=500   ;FEED RATE MILLING
Q351=+1    ;CLIMB OR UP-CUT
Q201=-20   ;DEPTH
Q202=5     ;PLUNGING DEPTH
Q369=0.1   ;ALLOWANCE FOR FLOOR
Q206=150   ;FEED RATE FOR PLNGNG
Q338=5     ;INFEED FOR FINISHING
Q200=2     ;SET-UP CLEARANCE
Q203=+0    ;SURFACE COORDINATE
Q204=50    ;2ND SET-UP CLEARANCE
Q366=1     ;PLUNGE
Q385=500   ;FINISHING FEED RATE
Q439=0     ;FEED RATE REFERENCE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

6.6 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 control 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 control performs a stepover with the current factor, and machines another revolution. The control takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached. If, on the other hand, you did not set the starting point on a side, but rather on a corner (**Q437** not equal to 0), the control mills on a spiral path from the starting point inward until the finished dimension has been reached.

5 If further stepovers are required, the tool is retracted from the contour on a tangential path and returns to the starting point of stud machining

6 The control 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 control positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If there is not enough room for the approach movement next to the stud, there is danger of collision.

- Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

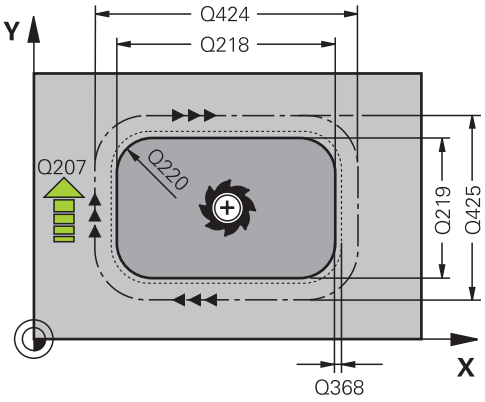
The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

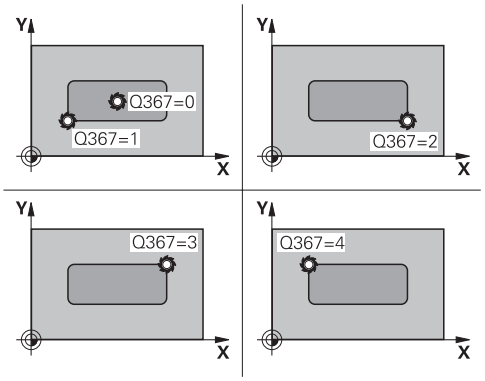
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

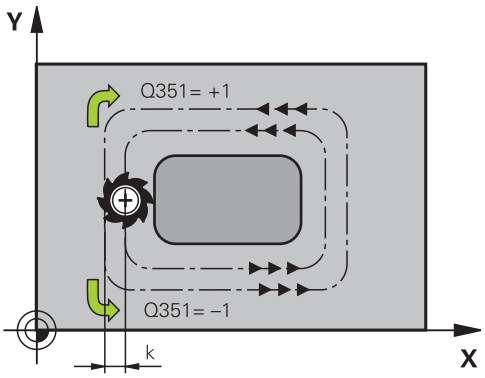
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

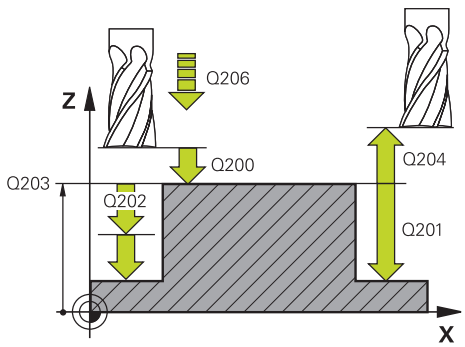
Cycle parameters

- **Q218 First side length?**: Stud length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q424 Workpiece blank side length 1?**: Length of the stud blank, parallel to the principal axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length**. The control performs multiple lateral 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 control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q219 Second side length?**: Stud length, parallel to the secondary axis of the working plane. Enter **Workpiece blank side length 2** greater than **Second side length**. The control performs multiple lateral 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 control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q425 Workpiece blank side length 2?**: Length of the stud blank, parallel to the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value between 0 and +99999.9999, the control rounds every corner. The value you enter here refers to the radius. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane, is left over after machining. Input range: 0 to 99999.9999
- **Q224 Angle of rotation?** (absolute): Angle by which the entire machining configuration is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: –360.0000 to 360.0000
- **Q367 Position of stud (0/1/2/3/4)?**: 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 = lower left corner
  2: Tool position = lower right corner
  3: Tool position = upper right corner
  4: Tool position = upper left corner









- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: –99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX** , **FAUTO**, **FU** , **FZ**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.1 to 1.9999; alternatively **PREDEF**

Example

```
8 CYCL DEF 256  RECTANGULAR STUD
    Q218=60      ;FIRST SIDE LENGTH
    Q424=74      ;WORKPC. BLANK SIDE 1
    Q219=40      ;2ND SIDE LENGTH
    Q425=60      ;WORKPC. BLANK SIDE 2
    Q220=5       ;CORNER RADIUS
    Q368=0.2     ;ALLOWANCE FOR SIDE
    Q224=+0      ;ANGLE OF ROTATION
    Q367=0       ;STUD POSITION
    Q207=500     ;FEED RATE MILLING
    Q351=+1      ;CLIMB OR UP-CUT
    Q201=-20     ;DEPTH
    Q202=5       ;PLUNGING DEPTH
    Q206=150     ;FEED RATE FOR PLNGNG
    Q200=2       ;SET-UP CLEARANCE
    Q203=+0      ;SURFACE COORDINATE
    Q204=50      ;2ND SET-UP CLEARANCE
    Q370=1       ;TOOL PATH OVERLAP
    Q437=0       ;APPROACH POSITION
    Q215=1       ;MACHINING OPERATION
    Q369=+0      ;ALLOWANCE FOR FLOOR
    Q338=+0      ;INFEED FOR FINISHING
    Q385=+0      ;FEED RATE FOR
                 FINISHING
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

- **Q437 Starting position (0...4)?**: Define the approach strategy of the tool:
  0: to the right of the stud (default setting)
  1: lower left corner
  2: lower right corner
  3: upper right corner
  4: upper left corner.
  If approach marks appear on the stud surface during approach with the setting **Q437**=0, then choose another approach position.
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**

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

Cycle run

Use Cycle 257 to machine a circular stud. The control 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 control 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 control moves the tool at rapid traverse **FMAX** to the set-up clearance **Q200**, and from there advances to the first plunging depth at the feed rate for plunging
4 The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5 The control 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:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

There is a danger of collision if there is insufficient room next to the stud.

- In this cycle, the control performs an approach movement
- To define the exact starting position, enter a starting angle between 0° and 360° in parameter **Q376**
- Depending on the starting angle **Q376**, the following amount of space must be left next to the stud: At least tool diameter +2 mm
- If you use the default value -1 the control automatically calculates the starting position

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

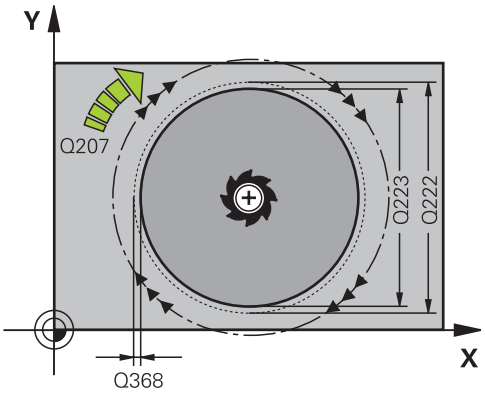
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

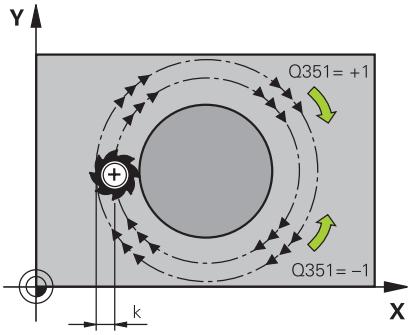
At the end of the cycle, the control returns the tool to the starting position.

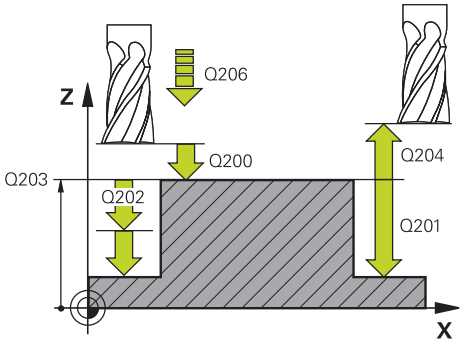
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

Cycle parameters

- **Q223 Finished part diameter?**: Diameter of the completely machined stud. Input range: 0 to 99999.9999
- **Q222 Workpiece blank diameter?**: Diameter of the workpiece blank. The workpiece blank diameter must be greater than the diameter of the finished part. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: –99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX** , **FAUTO**, **FU** , **FZ**







- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. Input range: 0.0001 to 1.9999; alternatively **PREDEF**
- **Q376 Starting angle?**: Polar angle relative to the stud center from where the tool approaches the stud. Input range: 0 to 359°
- **Q215 Machining operation (0/1/2)?**: Define the scope of machining:
  0: Roughing and finishing
  1: Roughing only
  2: Finishing only
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**

Example

```
8 CYCL DEF 257  CIRCULAR STUD
    Q223=60     ;FINISHED PART DIA.
    Q222=60     ;WORKPIECE BLANK DIA.
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q207=500    ;FEED RATE MILLING
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q206=150    ;FEED RATE FOR PLNGNG
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q370=1      ;TOOL PATH OVERLAP
    Q376=0      ;STARTING ANGLE
    Q215=+1     ;MACHINING OPERATION
    Q369=0      ;ALLOWANCE FOR FLOOR
    Q338=0      ;INFEED FOR FINISHING
    Q385=+500 ;FINISHING FEED RATE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

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

Cycle run

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

1 If the tool is below the 2nd set-up clearance at the beginning of machining, the control retracts the tool to the 2nd set-up clearance
2 Starting from the center of the stud the control moves the tool to the starting point of stud machining. The starting point depends, among other things, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter **Q224**
3 The tool moves at rapid traverse **FMAX** to the setup clearance **Q200** and from there with the feed rate for plunging to the first plunging depth.
4 The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5 The control 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 control returns the tool to the starting point of the stud milling process and then plunges the tool to the programmed depth
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 control will move the tool on the tool axis to the 2nd set-up clearance

Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

In this cycle, the control performs an automatic approach movement. If there is not enough space, a collision might occur.

- Use **Q224** to specify which angle is used to machine the first corner of the polygon stud. Input range: –360° to +360°
- Depending on the angle of rotation **Q224**, the following amount of space must be left next to the stud: At least tool diameter +2 mm

NOTICE

Danger of collision!

At the end, the control returns the tool 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 need not be the same as the starting position.

- Control the traversing movements of the machine
- In the simulation, control the end position of the tool after the cycle
- After the cycle, program the absolute (not incremental) coordinates

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

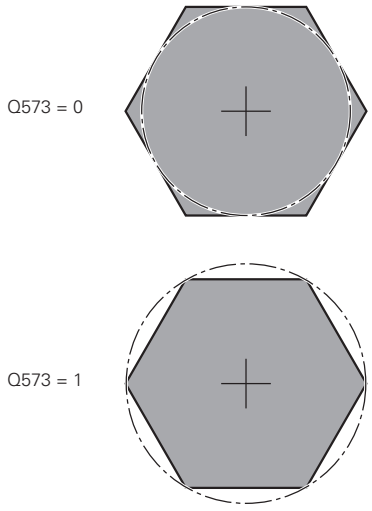
The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

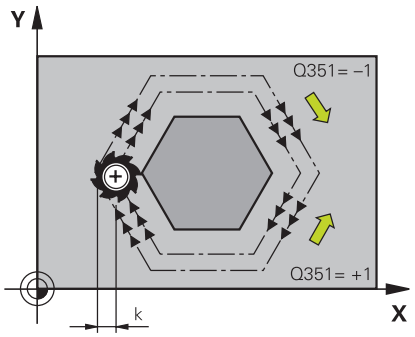
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

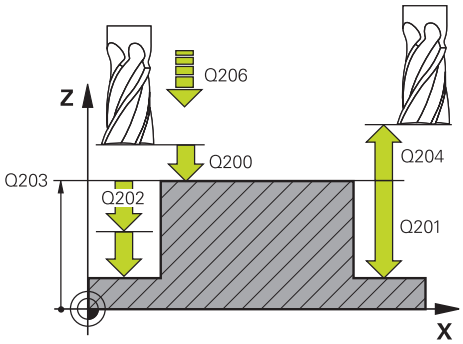
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

Cycle parameters

- **Q573 Inscr.circle/circumcircle (0/1)?**: Definition of whether the dimensioning shall reference the inscribed circle or the perimeter:
  0= dimensioning refers to the inscribed circle
  1= dimensioning refers to the perimeter
- **Q571 Reference circle diameter?**: Definition of the diameter of the reference circle. Specify in parameter **Q573** whether the diameter references the inscribed circle or the perimeter. Input range: 0 to 99999.9999
- **Q222 Workpiece blank diameter?**: Definition of the diameter of the workpiece blank. The workpiece blank diameter must be greater than the reference circle diameter. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q572 Number of corners?**: Enter the number of corners of the polygon stud. The control distributes the corners evenly on the stud. Input range: 3 to 30
- **Q224 Angle of rotation?**: Specify which angle is used to machine the first corner of the polygon stud. Input range: –360° to +360°
- **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value between 0 and +99999.9999, the control rounds every corner. The value you enter here refers to the radius. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. If you enter a negative value here, the control will return the tool to a diameter outside of the workpiece blank diameter after roughing. Input range: -99999.9999 to 99999.9999







- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: –99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX** , **FAUTO**, **FU** , **FZ**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

Example

```
8 CYCL DEF 258 POLYGON STUD
    Q573=1      ;REFERENCE CIRCLE
    Q571=50     ;REF-CIRCLE DIAMETER
    Q222=120    ;WORKPIECE BLANK DIA.
    Q572=10     ;NUMBER OF CORNERS
    Q224=40     ;ANGLE OF ROTATION
    Q220=2      ;RADIUS / CHAMFER
    Q368=0      ;ALLOWANCE FOR SIDE
    Q207=3000   ;FEED RATE MILLING
    Q351=1      ;CLIMB OR UP-CUT
    Q201=-18    ;DEPTH
    Q202=10     ;PLUNGING DEPTH
    Q206=150    ;FEED RATE FOR PLNGNG
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q370=1      ;TOOL PATH OVERLAP
    Q215=0      ;MACHINING OPERATION
    Q369=0      ;ALLOWANCE FOR FLOOR
    Q338=0      ;INFEED FOR FINISHING
    Q385=500    ;FINISHING FEED RATE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. Input range: 0.0001 to 1.9999; alternatively **PREDEF**
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**

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

Cycle run

With Cycle 233, you can 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

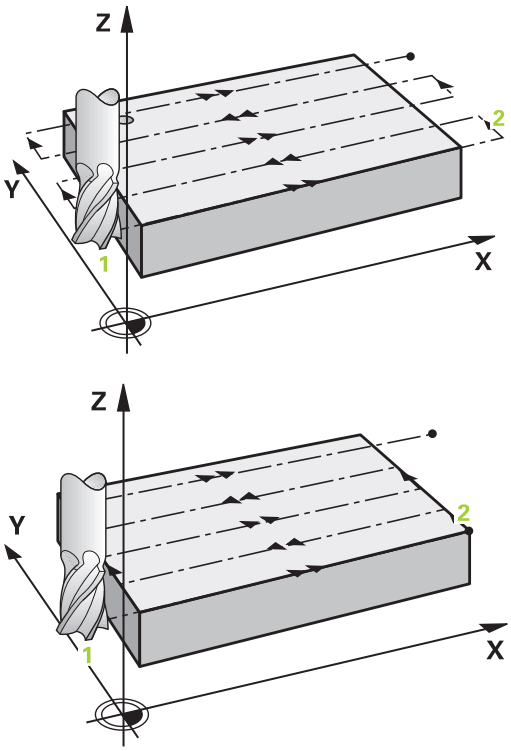
1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane: The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.

2 The control then positions the tool at rapid traverse **FMAX** to the set-up clearance in the spindle axis

3 The tool then moves in the tool axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control

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, with **Q389**=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389**=0 is used, the control additionally moves the tool beyond the level surface by the tool radius.



4 The control moves the tool to end point **2** at the programmed feed rate for milling

5 Then the control 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 set-up clearance to the side

6 The tool then returns in the opposite direction at the feed rate for milling

7 The process is repeated until the programmed surface has been completed.

8 The control then positions the tool at rapid traverse **FMAX** back to starting point **1**

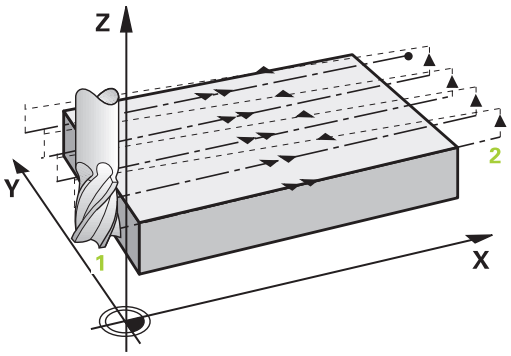
9 If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate

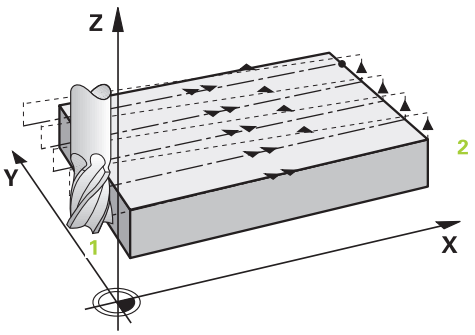
10 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate

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, with **Q389**=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389**=2 is used, the control additionally moves the tool beyond the level surface by the tool radius.





4 The tool subsequently advances at the programmed feed rate for milling to end point **2**

5 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves at **FMAX** directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side.

6 The tool then returns to the current infeed depth and moves in the direction of end point **2**

7 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**

8 If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate

9 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate

10 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.

Strategy Q389=4

4 The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a tangential arc

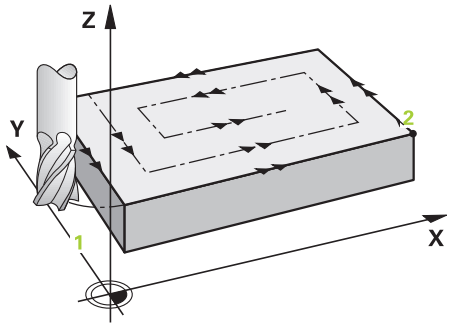
5 The control machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged

6 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**

7 If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate

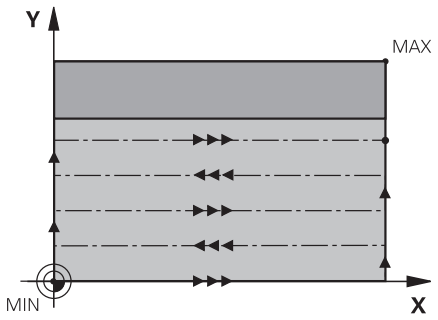
8 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate

9 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.



Limits

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



Please note while programming:

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

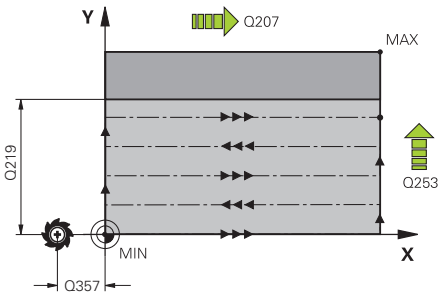
If you define **Q370 TOOL PATH OVERLAP** >1, the programmed overlap factor will be taken into account right from the first machining path.

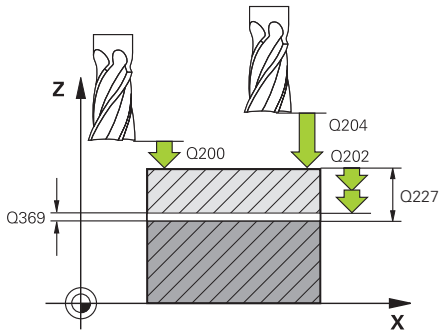
Cycle 233 monitors the entries made for the tool or cutting edge length in **LCUTS** in the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.

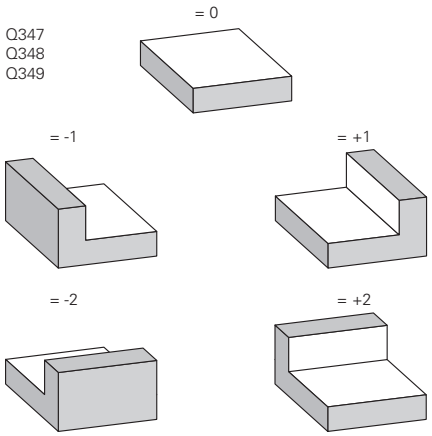
If a limit (**Q347**, **Q348** or **Q349**) was programmed in the machining direction **Q350**, the cycle will extend the contour in the infeed direction by corner radius **Q220**. The specified surface will be machined completely.

Cycle parameters

- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q389 Machining strategy (0-4)?**: Determine how the control 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 outside the surface being machined
  3: Line-by-line machining, retraction and stepover at the positioning feed rate at the edge of the surface being machined
  4: Helical machining, uniform infeed from the outside toward the inside
- **Q350 Milling direction?**: Axis in the working plane that defines the machining direction:
  1: Principal axis = machining direction
  2: Secondary axis = machining direction
- **Q218 First side length?** (incremental): Length of the surface to be machined in the principal axis of the working plane, referencing the starting point in the 1st axis. Input range: -99999.9999 to 99999.9999
- **Q219 Second side length?** (incremental): Length of the surface to be machined in the secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referencing the **STARTNG PNT 2ND AXIS**. Input range: –99999.9999 to 99999.9999







- **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range: –99999.9999 to 99999.9999
- **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to which the surface is to be face-milled. Input range: –99999.9999 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range: 0 to 99999.9999
- **Q202 MAX. PLUNGING DEPTH** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q370 Path overlap factor?**: Maximum stepover factor k. The control 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 to 1.9999.
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range: 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (**Q389**=1), the control uses the cross feed rate for milling **Q207**. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q357 Safety clearance to the side?** (incremental) Parameter **Q357** influences the following situations:
  Approaching the first plunging depth: **Q357** is the lateral distance from the tool to the workpiece
  Roughing with the milling strategies Q389=0 to 3: The surface to be machined in the **Q350 MILLING DIRECTION** is increased by the value from **Q357** if there is no limit set in this direction
  Side finishing: The paths are extended by the value in **Q357** in the **Q350 MILLING DIRECTION**
  Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

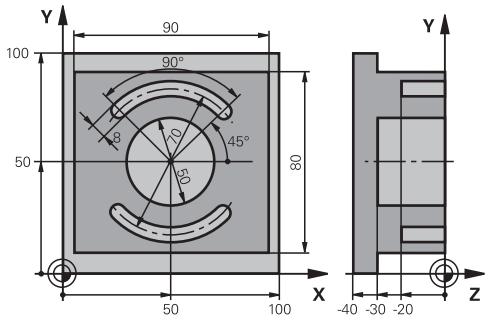
Example

```
8 CYCL DEF 233  FACE MILLING
    Q215=0      ;MACHINING OPERATION
    Q389=2      ;MILLING STRATEGY
    Q350=1      ;MILLING DIRECTION
    Q218=120    ;FIRST SIDE LENGTH
    Q219=80     ;2ND SIDE LENGTH
    Q227=0      ;STARTNG PNT 3RD AXIS
    Q386=-6     ;END POINT 3RD AXIS
    Q369=0.2    ;ALLOWANCE FOR FLOOR
    Q202=3      ;MAX. PLUNGING DEPTH
    Q370=1      ;TOOL PATH OVERLAP
    Q207=500    ;FEED RATE MILLING
    Q385=500    ;FINISHING FEED RATE
    Q253=750    ;F PRE-POSITIONING
    Q357=2      ;CLEARANCE TO SIDE
    Q200=2      ;SET-UP CLEARANCE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q347=0      ;1ST LIMIT
    Q348=0      ;2ND LIMIT
    Q349=0      ;3RD LIMIT
    Q220=2      ;CORNER RADIUS
    Q368=0      ;ALLOWANCE FOR SIDE
    Q338=0      ;INFEED FOR FINISHING
    Q367=-1     ;SURFACE POSITION
                (-1/0/1/2/3/4)?
9 L  X+0 Y+0  R0 FMAX  M3 M99
```

- **Q347 1st limit?**: Select the side of the workpiece where the level surface is bordered by a side wall (not possible with helical machining). Depending on the position of the side wall, the control limits the machining of the plane surface to the corresponding starting point coordinate or side length: (not possible with helical machining):
  Input **0**: No limiting
  Input **–1**: Limit in negative principal axis
  Input **+1**: Limiting in positive principal axis
  Input **–2**: Limiting in negative secondary axis
  Input **+2**: Limiting in positive secondary axis
- **Q348 2nd limit?**: See parameter 1st limit **Q347**
- **Q349 3rd limit?**: See parameter 1st limit **Q347**
- **Q220 Corner radius?**: Radius of a corner at limits (**Q347** to **Q349**). Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q367 Surface position (-1/0/1/2/3/4)?**: Position of the surface with respect to the position of the tool when the cycle is called:
  –1: Tool position = current position
  0: Tool position = stud center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner

6.10 Programming Examples

Example: Milling pockets, studs and slots



0 BEGINN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call: roughing/finishing
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Cycle definition: outside machining
Q218=90 ;FIRST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q220=0 ;CORNER RADIUS	
Q368=0 ;ALLOWANCE FOR SIDE	
Q224=0 ;ANGLE OF ROTATION	
Q367=0 ;STUD POSITION	
Q207=250 ;FEED RATE MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q437=0 ;APPROACH POSITION	
6 L X+50 Y+50 R0 M3 M99	Cycle call for outside machining
7 CYCL DEF 252 CIRCULAR POCKET	Cycle definition: circular pocket
Q215=0 ;MACHINING OPERATION	
Q223=50 ;CIRCLE DIAMETER	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q207=500 ;FEED RATE MILLING	

Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q366=1 ;PLUNGE	
Q385=750 ;FINISHING FEED RATE	
Q439=0 ;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99	Cycle call for circular pocket
9 TOOL CALL 2 Z S5000	Tool call: slot milling cutter
10 CYCL DEF 254 CIRCULAR SLOT	Cycle definition: slots
Q215=0 ;MACHINING OPERATION	
Q219=8 ;SLOT WIDTH	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q375=70 ;PITCH CIRCLE DIAMETR	
Q367=0 ;REF. SLOT POSITION	No pre-positioning in X/Y required
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q376=+45 ;STARTING ANGLE	
Q248=90 ;ANGULAR LENGTH	
Q378=180 ;STEPPING ANGLE	Starting point for second slot
Q377=2 ;NR OF REPETITIONS	
Q207=500 ;FEED RATE MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-20 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGE	
Q385=500 ;FINISHING FEED RATE	
Q439=0 ;FEED RATE REFERENCE	
11 CYCL CALL FMAX M3	Cycle call for slots
12 L Z+250 R0 FMAX M2	Retract the tool, end program
13 END PGM C210 MM	

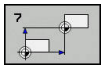
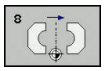
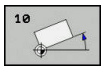
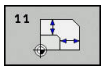
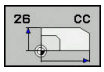
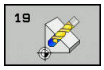
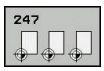
7

Cycles: Coordinate Transformations

7.1 Fundamentals

Overview

Once a contour has been programmed, the control can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The control provides the following functions for coordinate transformations:

Soft key	Cycle	Page
	7 DATUM SHIFT For shifting contours directly within the NC program or from datum tables	219
	8 MIRRORING Mirroring contours	226
	10 ROTATION Rotating contours in the working plane	228
	11 SCALING FACTOR Increasing or reducing the size of contours	230
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	231
	19 WORKING PLANE Machining in a tilted coordinate system on machines with swivel heads and/or rotary tables	233
	247 PRESETTING Presetting during the program run	240

Effectiveness of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

Reset coordinate transformation:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

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

Effect

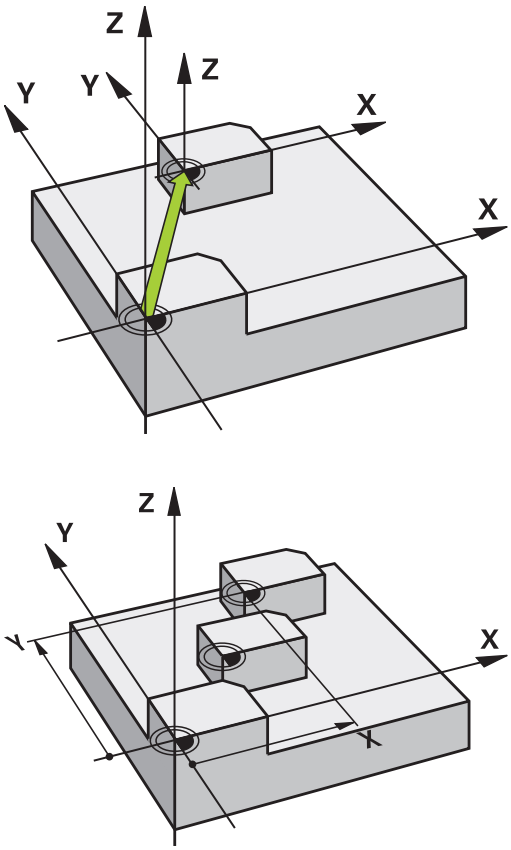
Refer to your machine manual!

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

After the definition of a datum shift cycle, all coordinate data will reference the new datum. The control displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

Resetting

- To shift the datum back to the coordinates X=0, Y=0 etc., program another cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.



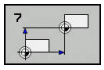
Please note while programming

Possible datum shift values in the rotary axes will be specified by your machine tool builder in the **presetToAlignAxis** parameter (no. 300203).

In the machine parameter **CfgDisplayCoordSys** (no. 127501), the machine tool builder specifies the coordinate system in which the status display shows an active datum shift.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Cycle parameters



- **Displacement**: Enter the coordinates of the new datum. Absolute values reference the workpiece datum defined by presetting. Incremental values always reference 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

Example

```
13 CYCL DEF 7.0  DATUM SHIFT
14 CYCL DEF 7.1  X+60
15 CYCL DEF 7.2  Y+40
16 CYCL DEF 7.3  Z-5
```

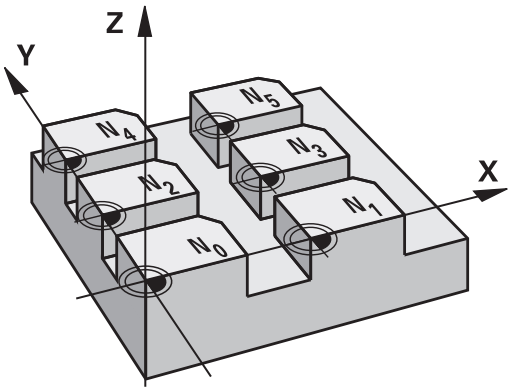
7.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

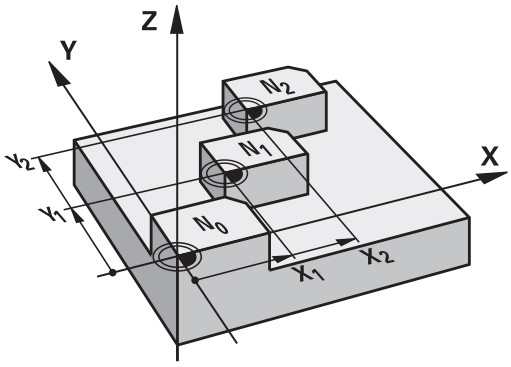
Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within an NC 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.
- To shift the datum back to the coordinates X=0, Y=0 etc., directly call a cycle definition

Status displays

In the additional status display, the following data from the datum table is 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:

In the machine parameter **CfgDisplayCoordSys** (no. 127501), the machine tool builder specifies the coordinate system in which the status display shows an active datum shift.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Datums from a datum table **always and exclusively** reference the current 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 program run):

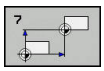
- Use the file manager to select the desired table for a test run in the **Test Run** operating mode: The table now has the status S
- Use the file manager in the **Program run, single block** and **Program run, full sequence** operating modes to select the desired table for program run: The table receives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

The coordinate values from datum tables are only effective with absolute coordinate values.

New lines can only be inserted at the end of the table.

If you create datum tables, the file name has to start with a letter.

Cycle parameters



- **Displacement**: Enter the number of the datum from the datum table or in a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input range: 0 to 9999

Example

```
77 CYCL DEF 7.0  DATUM SHIFT
78 CYCL DEF 7.1  #5
```

Selecting a datum table in the part program

With the **SEL TABLE** function, you select the datum table from which the control takes the datums:

Proceed as follows:

PGM CALL

- Press the **PGM CALL** key

SELECT DATUM TABLE

- Press the **SELECT DATUM TABLE** soft key
- Enter the complete path name of the datum table

SELECT FILE

- Alternative: Press the **SELECT FILE** soft key
- Confirm your input with the **END** key.

> Program a **SEL TABLE** block before Cycle 7 Datum Shift.
>
> A datum table selected with **SEL TABLE** remains active until you select another datum table with **SEL TABLE** or through **PGM MGT**.

Editing the datum table in the Programming mode of operation

> After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise, the change will not be taken into account when the NC program is executed.

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

Proceed as follows:

PGM MGT

- Press the **PGM MGT** key

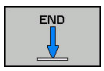
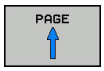
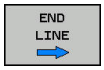
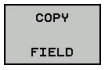
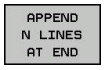
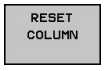
SELECT TYPE

- Press the **SELECT TYPE** soft key

SHOW ALL

- Press the **SHOW ALL** soft key
- Select the desired table or enter a new file name
- Select the file with the **ENT** key

The functions in the soft-key row include:

Soft key	Function
	Select the beginning of the table
	Select the table end
	Go to previous page
	Go to next page
	Find (a small window opens where you can enter the text or value you are looking for)
	Reset table
	Move the cursor to the beginning of the line
	Move the cursor to the end of the line
	Copy the current value
	Insert the copied value
	Add the entered number of lines (datums) to the end of the table
	Insert line (only possible at the end of table)
	Delete line
	Sort or hide columns (a window opens)
	Displays more functions: Delete, select, deselect, save as
	Reset the column
	Edit the current field
	Sort the datums (a window opens where you can select the sorting order)

Editing the datum table in the Program Run, Single Block or Program Run, Full Sequence operating mode

Select the datum table in the **Program Run, Full Sequence/Single Block** operating mode.

Proceed as follows:

- Shift the soft-key row

- Press the **SELECT COMPENS. TABLES** soft key

- Press the **DATUM TABLE** soft key

To capture the actual position and store it in the datum table, proceed as follows:

- Set the **EDIT** soft key to **ON**
- Use the arrow keys to navigate to the desired source

- Press the **ACTUAL POSITION CAPTURE** key

> The control only captures the actual position for the axis in which the cursor is currently positioned.

After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise, the change will not be taken into account when the NC program is executed.

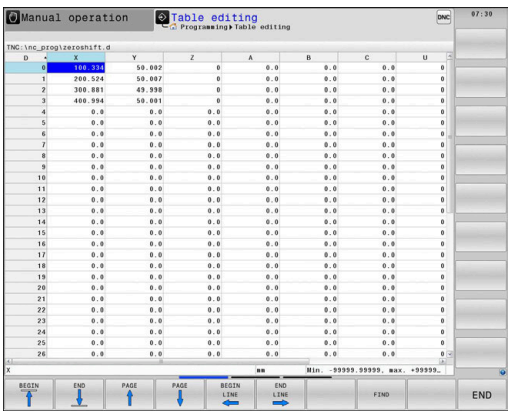
The change of a datum will only become effective after you have called Cycle 7 again.

While an NC program is running, you cannot access the datum table. For compensations during the program run, use the **COMPENS. TABLE T-CS** or **COMPENS. TABLE WPL-CS** soft key.
Further information: User's Manual for Conversational Programming

Configuring a datum table

If you do not wish to define a datum for an active axis, press the **DEL** key. Then the control clears the numerical value from the corresponding input field.

You can change the properties of tables. Enter code number 555343 in the MOD menu. The control then displays the **EDIT FORMAT** soft key if a table is selected. When you press this soft key, the control opens a pop-up window where the properties are shown for each column of the selected table. Any changes you make only affect the open table.



Leaving a datum table

Select a different type of file in file management. Select the desired file.

NOTICE

Danger of collision!

The control considers changes in a datum table only when the values are saved.

- Make sure to confirm any changes made to the table immediately by pressing the **ENT** key
- Carefully test the NC program after making a change to the datum table

Status displays

In the additional status display, the control shows the values of the active datum shift.

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

Effect

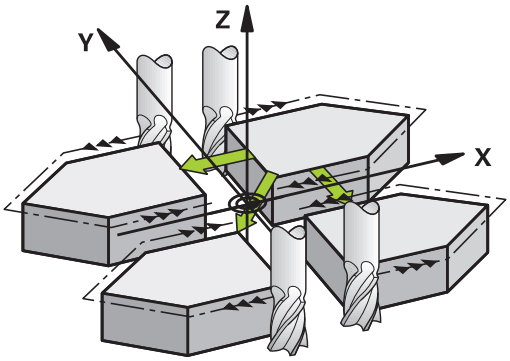
The control can machine the mirror image of a contour in the working plane.

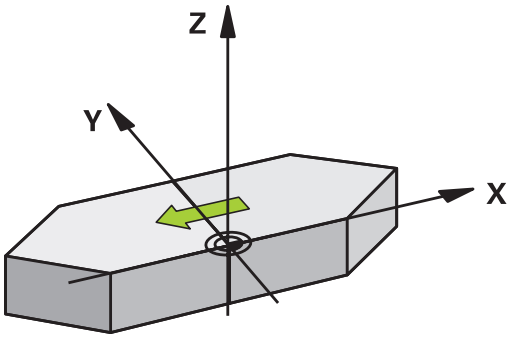
Mirroring becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed; this does not apply to 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**.

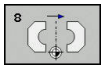
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

Cycle parameters



- **Mirror image axis?**: Enter the axis to be mirrored. You can mirror all axes—including rotary axes—except for the spindle axis and its associated secondary axis. You can enter up to three axes. Input range: up to three NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

```
79 CYCL DEF 8.0  MIRRORING
80 CYCL DEF 8.1  X Y  Z
```

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

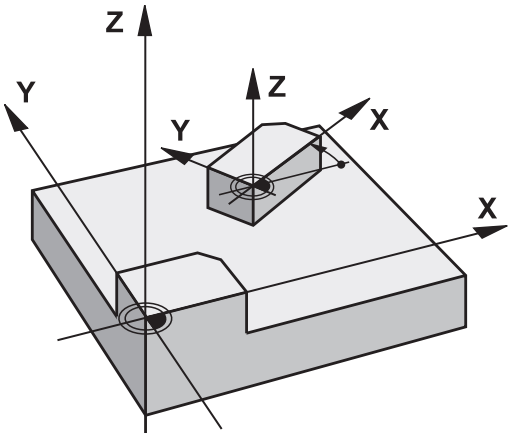
Effect

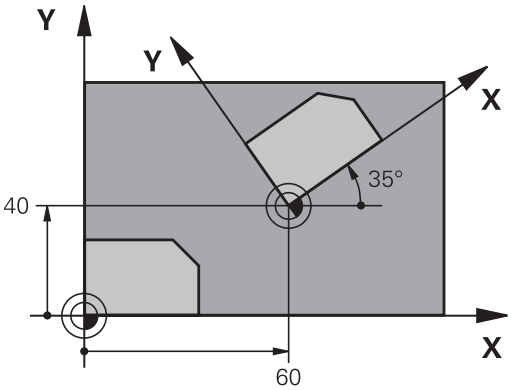
Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

The ROTATION cycle becomes effective as soon as it has been defined in the NC program. It is also effective in the Positioning with Manual Data Input operating mode. The active angle of rotation 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°.

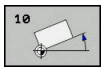
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle 10 cancels an active radius compensation. If necessary, reprogram the radius compensation.

After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

Cycle parameters



- **Rotation**: Enter the angle of rotation in degrees (°). Input range: -360.000° to +360.000° (absolute or incremental)

Example

```
12 CALL LBL 1
13 CYCL DEF 7.0  DATUM SHIFT
14 CYCL DEF 7.1  X+60
15 CYCL DEF 7.2  Y+40
16 CYCL DEF 10.0  ROTATION
17 CYCL DEF 10.1  ROT+35
18 CALL LBL 1
```

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

Effect

The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

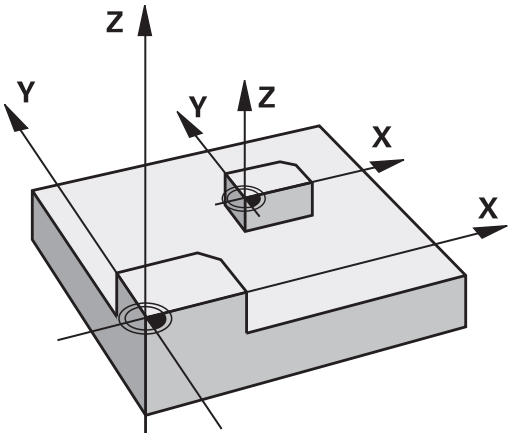
- all three coordinate axes at the same time
- dimensions in cycles

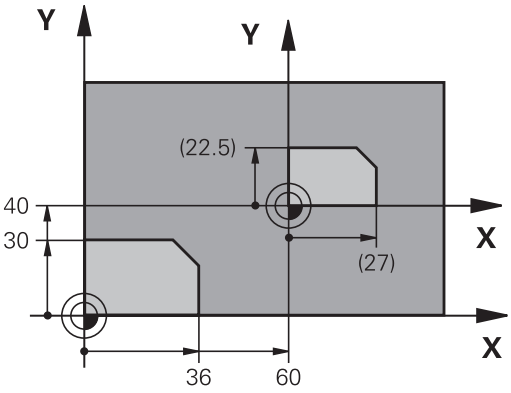
Prerequisite

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)



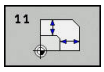


> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Resetting

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

Cycle parameters



- **Factor?**: Enter the scaling factor SCL. The control multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range: 0.000001 to 99.999999

Example

```
11 CALL LBL 1
12 CYCL DEF 7.0  DATUM SHIFT
13 CYCL DEF 7.1  X+60
14 CYCL DEF 7.2  Y+40
15 CYCL DEF 11.0  SCALING
16 CYCL DEF 11.1  SCL 0.75
17 CALL LBL 1
```

7.7 AXIS-SPECIFIC SCALING (Cycle 26)

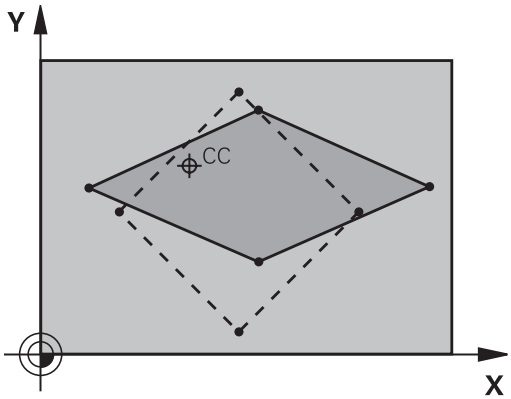
Effect

With Cycle 26, you can account for shrinkage and oversize factors for each axis.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. 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 corresponding axis.



Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

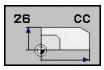
Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

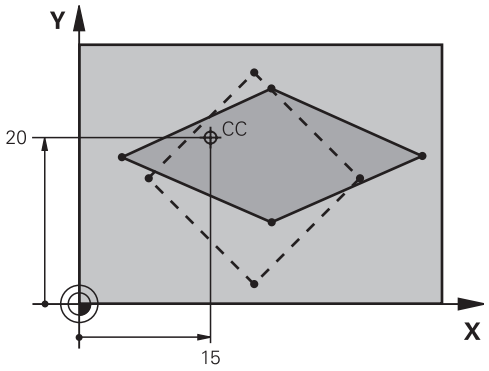
In addition, you can enter the coordinates of a center for all scaling factors.

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 factor**: Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction. 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



Example

```
25 CALL LBL 1
26 CYCL DEF 26.0  AXIS-SPECIFIC
   SCALING
27 CYCL DEF 26.1  X 1.4  Y 0.6  CCX+15
   CCY+20
28 CALL LBL 1
```

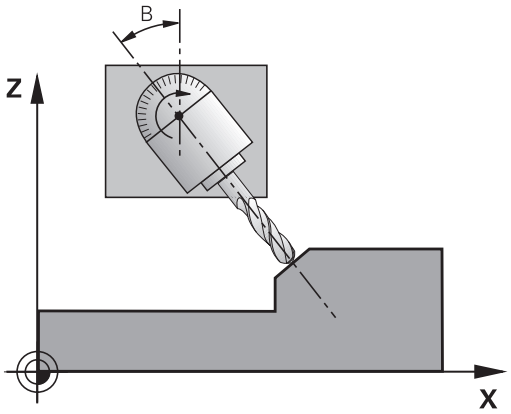
7.8 WORKING PLANE (Cycle 19, DIN/ISO: G80, option 1)

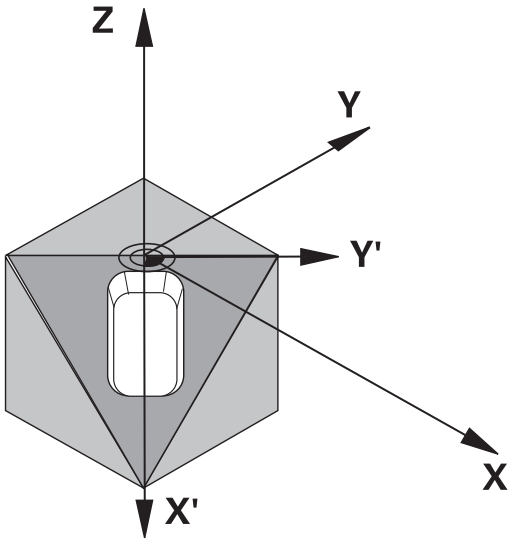
Effect

In Cycle 19 you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to three rotations (spatial angles) of the **machine-based** coordinate system.

  The required spatial angles 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 control will calculate the required angle positions of the tilted axes automatically and will store these in the **Q120** (A axis) to **Q122** (C axis) parameters. If two solutions are possible, the control will choose the shorter path from the current position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The control first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it has been defined in the NC program. As soon as you move an axis in the tilted system, the compensation for this specific axis will be activated. You must move all axes to activate compensation for all axes.

If you set the **Tilting program run** parameters 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 **Tilt working plane** functions are adapted to the control and the machine tool by the machine tool builder.

The machine tool builder also specifies whether the programmed angles are interpreted as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).

In the machine parameter **CfgDisplayCoordSys** (no. 127501), the machine tool builder specifies the coordinate system in which the status display shows an active datum shift.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

This cycle can also be used in the **FUNCTION MODE TURN** machining mode if this mode is executed using radial facing slide kinematics.

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.

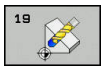
The working plane is always tilted around the active datum.

If you use the Cycle 19 while M120 is active, the control automatically cancels the radius compensation, which also cancels the M120 function.

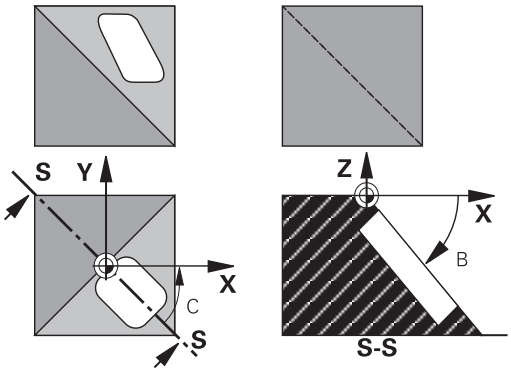
Write the program as if the machining process was to be executed in a non-tilted plane.

If you call the cycle again for other angles, you do not need to reset the machining parameters.

Cycle parameters



- **Rotary axis and angle?**: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range: –360.000 to 360.000



If the control automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate? F=**: Traversing speed of the rotary axis during automatic positioning. Input range 0 to 99999.999
- **Set-up clearance?** (incremental): The control positions the tilting head in such a way that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range: 0 to 99999.9999

Reset

To reset the tilt angles, redefine the WORKING PLANE cycle. Enter an angular value of 0° for all rotary axes. Then, redefine the WORKING PLANE cycle. Confirm the dialog prompt by pressing the **NO ENT** key. This disables the function.

Positioning the axes of rotation

Refer to your machine manual!

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

Manual positioning of rotary axes

If Cycle 19 does not position the rotary axes automatically, you need to position them in a separate L block following 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 program the Q parameters **Q120** (A axis value), **Q121** (B axis value) and **Q122** (C axis value) according to 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

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the spatial angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0	
14 L A+Q120 C+Q122 R0 F1000	Position the rotary axes by using values calculated by Cycle 19
15 L Z+80 R0 FMAX	Activate compensation for the spindle axis
16 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle 19:

- The control can position only closed-loop axes.
- To position the tilted axes, you must enter a feed rate and a set-up clearance, in addition to the tilting angles, when defining the cycle
- Use only preset tools (the full tool length must have been defined)
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting.
- The control performs tilting at the last programmed feed rate (the maximum feed rate depends on the complexity of the swivel head geometry or tilting table)

Example

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0 F5000 ABST50	Also define the feed rate and the clearance
14 L Z+80 R0 FMAX	Activate compensation for the spindle axis
15 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Position display in a tilted system

On activation of Cycle 19, the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Monitoring of the working space

The control monitors only those axes in the tilted coordinate system that are moved. Where applicable, the control displays 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.

It is also possible to position the axes using straight-line blocks that reference the machine coordinate system (NC blocks with M91 or M92), if the working plane is tilted. Restrictions:

- Positioning is without length compensation.
- Positioning is done without length compensation.
- Tool radius compensation is not allowed.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is tilted about the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

1st: Activate datum shift
2nd: Activate **Tilt working plane**
3rd: Activate rotation
...
Workpiece machining
...
1st: Reset rotation
2nd: Reset **Tilt working plane**
3rd: Reset datum shift

Procedure for working with Cycle 19 WORKING PLANE

Proceed as follows:

- Create the NC program
- Clamp the workpiece
- Set any presets
- Start the NC program

Creating the NC program:

- Call the defined tool
- Retract in the spindle axis
- Position the axes of rotation
- Activate a datum shift if required
- Define Cycle 19 **WORKING PLANE**
- Position all principal axes (X, Y, Z) in order to activate the compensation
- Define Cycle 19 with other angles, if required
- Reset Cycle 19 by programming 0° for all tilt axes
- In order to deactivate the working plane, define Cycle 19 again
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

You can define the preset in the following ways:

- Manually by touch-off
- Controlled with a HEIDENHAIN 3-D touch probe
- Automatically with a HEIDENHAIN 3-D touch probe

Further information: User's Manual for Setup, Testing and Running NC Programs
Further information: "Touch Probe Cycles: Automatic Presetting", Page 647

7.9 PRESETTING (Cycle 247, DIN/ISO: G247)

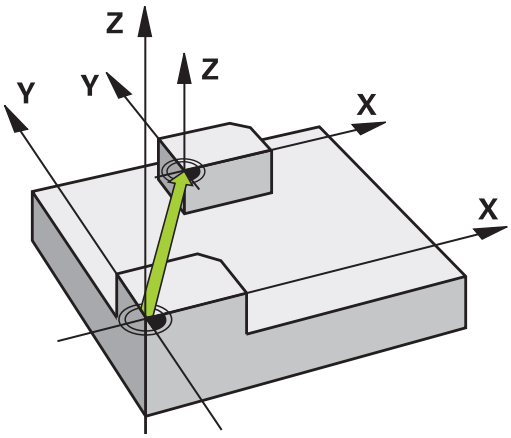
Effect

With the presetting cycle you can activate as the new preset a preset defined in the preset table.

After a presetting 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 control shows the active preset number behind the preset symbol.



Please note before programming:

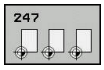
This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

When activating a preset from the preset table, the control 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 preset that you last set in the **Manual operation** or **Electronic handwheel** operating mode.

Cycle 247 is also effective in the Test Run operating mode.

Cycle parameters



- **Number for preset?**: Enter the number of the desired preset from the preset table. Alternatively, you can press the **SELECT** soft key and directly select the desired preset from the preset table. Input range: 0 to 65535

Example

```
13 CYCL DEF 247  PRESETTING
    Q339=4       ;PRESET NUMBER
```

Status displays

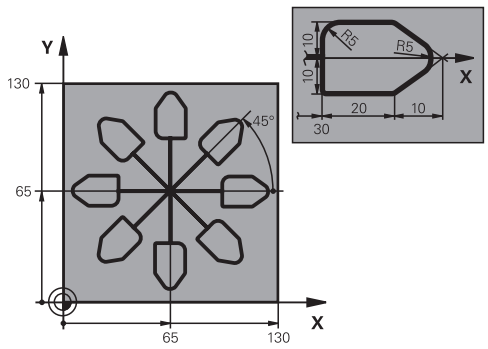
In the additional status display (**STATUS POS.**) the control shows the active preset number in addition to the **Datum** dialog.

7.10 Programming Examples

Example: coordinate transformation cycles

Program run

- Program the coordinate transformations in the main program
- Machining within a subprogram



0 BEGIN PGM COTRANS MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+130 X+130 Z+0	
3 TOOL CALL 1 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
6 CYCL DEF 7.1 X+65	
7 CYCL DEF 7.2 Y+65	
8 CALL LBL 1	Call milling operation
9 LBL 10	Set label for program section repeat
10 CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
11 CYCL DEF 10.1 IROT+45	
12 CALL LBL 1	Call milling operation
13 CALL LBL 10 REP 6/6	Return jump to LBL 10; repeat the milling operation six times
14 CYCL DEF 10.0 ROTATION	Reset the rotation
15 CYCL DEF 10.1 ROT+0	
16 CYCL DEF 7.0 DATUM SHIFT	Reset datum shift
17 CYCL DEF 7.1 X+0	
18 CYCL DEF 7.2 Y+0	
19 L Z+250 R0 FMAX M2	Retract the tool, end program
20 LBL 1	Subprogram 1
21 L X+0 Y+0 R0 FMAX	Define milling operation
22 L Z+2 R0 FMAX M3	
23 L Z-5 R0 F200	
24 L X+30 RL	
25 L IY+10	
26 RND R5	
27 L IX+20	
28 L IX+10 IY-10	

```
29 RND  R5
30 L  IX-10 IY-10
31 L  IX-20
32 L  IY+10
33 L  X+0 Y+0  R0 F5000
34 L  Z+20 R0  FMAX
35 LBL 0
36 END PGM  COTRANS MM
```

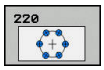
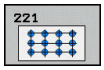
8

Fixed Cycles: Pattern definitions

8.1 Fundamentals

Overview

The control provides three cycles for machining point patterns:

Soft key	Cycle	Page
	220 POLAR PATTERN	246
	221 CARTESIAN PATTERN	249
	224 DATAMATRIX CODE PATTERN	251

You can combine Cycles 220, 221, and 224 with the following fixed cycles:

Cycle 200 **DRILLING**
Cycle 201 **REAMING**
Cycle 203 **UNIVERSAL DRILLING**
Cycle 205 **UNIVERSAL PECKING**
Cycle 208 **BORE MILLING**
Cycle 240 **CENTERING**
Cycle 251 **RECTANGULAR POCKET**
Cycle 252 **CIRCULAR POCKET**

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

Cycle 202 **BORING**
Cycle 204 **BACK BORING**
Cycle 206 **TAPPING**
Cycle 207 **RIGID TAPPING**
Cycle 209 **TAPPING W/ CHIP BRKG**
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 MLLNG/CNTSNKG**
Cycle 264 **THREAD DRILLNG/MLLNG**
Cycle 265 **HEL. THREAD DRLG/MLG**
Cycle 267 **OUTSIDE THREAD MLLNG**

If you have to machine irregular point patterns, use **CYCL CALL PAT** to develop point tables.

More regular point patterns are available with the **PATTERN DEF** function.

Further information: "Point tables", Page 79

Further information: "Pattern definition with PATTERN DEF", Page 72

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

Cycle run

1 The control moves the tool at rapid traverse from its current position to the starting point for the first machining operation.

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
- Approach the starting point in the spindle axis.
- Move to the set-up clearance above the workpiece surface (spindle axis)

2 From this position, the control executes the last defined fixed machining cycle

3 The tool then approaches the starting point for the next machining operation on a straight lineor a circular arc. 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:

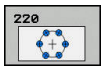
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle 220 is DEF-active. In addition, Cycle 220 automatically calls the last defined fixed cycle.

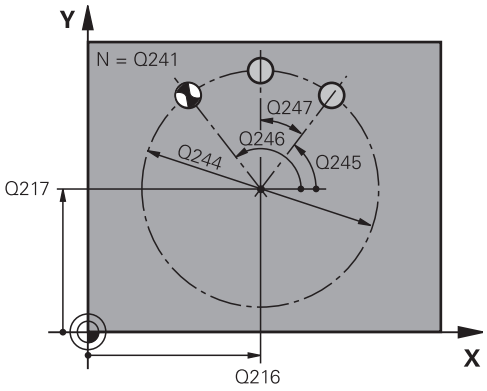
If you combine Cycle 220 or Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 or 221 will be effective. This applies within the NC program until the affected parameters are overwritten again. Example: If, in an NC program Cycle 200 is defined with **Q203**=0 and you then program a Cycle 220 with **Q203**=–5, then the subsequent calls with **CYCL CALL** and **M99** will use **Q203**=-5. Cycles 220 and 221 overwrite the above-mentioned parameters of **CALL**-active machining cycles (if the same input parameters have been programmed in both cycles).

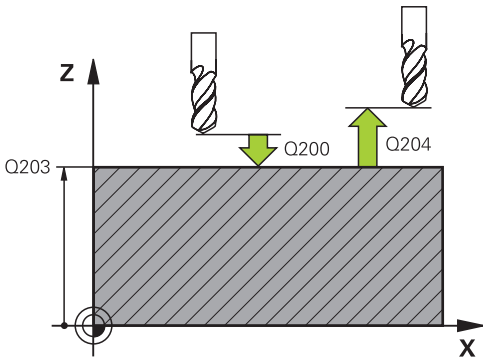
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q216 Center in 1st axis?** (absolute): Pitch circle center in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range: 0 to 99999.9999
- **Q245 Starting angle?** (absolute): Angle between the principal axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range: –360.000 to 360.000
- **Q246 Stopping angle?** (absolute): Angle between the principal axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you specify a stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range: –360.000 to 360.000
- **Q247 Intermediate stepping angle?** (incremental): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the control 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 control will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). Input range: –360.000 to 360.000
- **Q241 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range: 1 to 99999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999





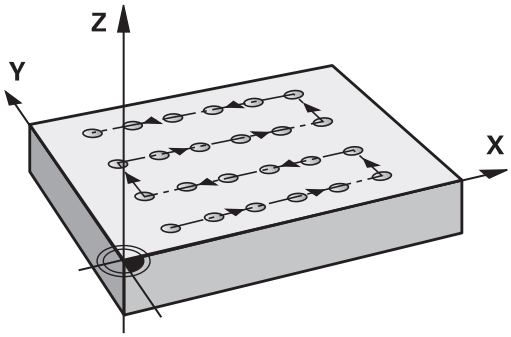
Example

```
53 CYCL DEF 220  POLAR PATTERN
    Q216=+50    ;CENTER IN 1ST AXIS
    Q217=+50    ;CENTER IN 2ND AXIS
    Q244=80     ;PITCH CIRCLE DIAMETR
    Q245=+0     ;STARTING ANGLE
    Q246=+360   ;STOPPING ANGLE
    Q247=+0     ;STEPPING ANGLE
    Q241=8      ;NR OF REPETITIONS
    Q200=2      ;SET-UP CLEARANCE
    Q203=+30    ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q301=1      ;MOVE TO CLEARANCE
    Q365=0      ;TYPE OF TRAVERSE
```

- **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
  0: Move to set-up clearance between machining operations
  1: Move to 2nd set-up clearance between machining operations
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool will move 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

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

Cycle run



1 The control automatically moves the tool from its current position to the starting point for the first machining operation

   Sequence:

   - Move to the 2nd set-up clearance (spindle axis)
   - Approach the starting point in the machining plane
   - Move to the set-up clearance above the workpiece surface (spindle axis)

2 From this position, the control executes the last defined fixed machining cycle

3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance)

4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first line have been completed. The tool is located above the last point of 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 direction of the reference axis.

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:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

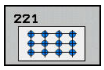
Cycle 221 is DEF-active. In addition, Cycle 221 automatically calls the last defined fixed cycle.

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

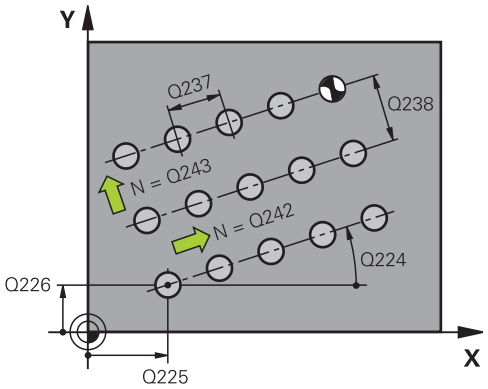
The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

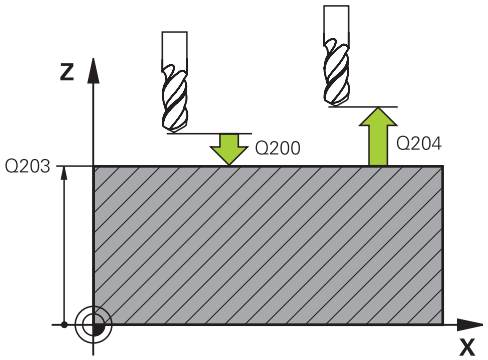
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



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





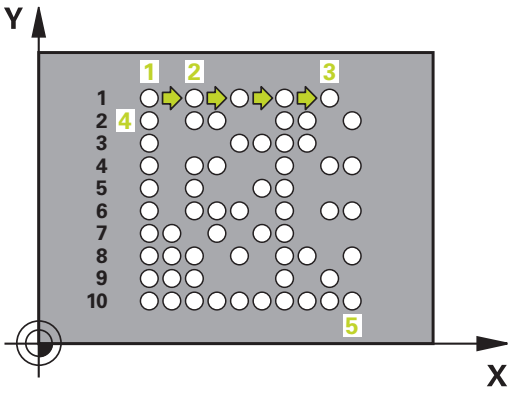
Example

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

8.4 DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224)

Cycle run

With Cycle 224 **DATAMATRIX CODE PATTERN**, you can convert text to a so-called DataMatrix code. This code will be used as a point pattern for a previously defined fixed cycle.



1 The control automatically moves the tool from its current position to the programmed starting point. This point is always located in the lower left corner.

   Sequence:

   - Move to the 2nd set-up clearance (spindle axis)
   - Approach the starting point in the machining plane
   - Move to the Safety clearance above the workpiece surface (spindle axis)

2 Then, the control moves the tool in the positive direction of the secondary axis to the first starting point **1** in the first row

3 From this position, the control executes the last defined fixed machining cycle

4 Then, the control moves the tool in the positive direction of the principal axis to the second starting point **2** of the next machining operation. The tool stops at the 1st set-up clearance

5 This procedure will be repeated until all machining operations in the first row have been completed. The tool is located above the last point **3** of the first row

6 Then, the control moves the tool in the negative direction of the principal and secondary axes to the first starting point **4** of the next row

7 Then, the next points are machined

8 These steps are repeated until the entire DataMatrix code has been completed. Machining stops in the lower right corner **5**

9 Finally, the control retracts the tool to the programmed 2nd set-up clearance

Please note while programming!

NOTICE

Danger of collision!

If you combine Cycle 224 with one of the fixed cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle 224 will be effective for the selected fixed cycle.

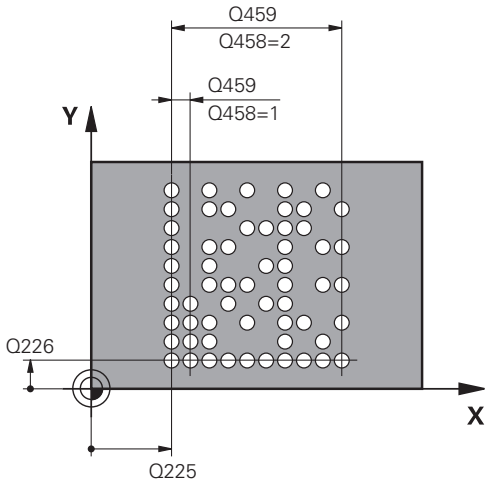
- Check the machining sequence using a graphic simulation
- Carefully test the NC program or program section in **Program run, single block** operating mode

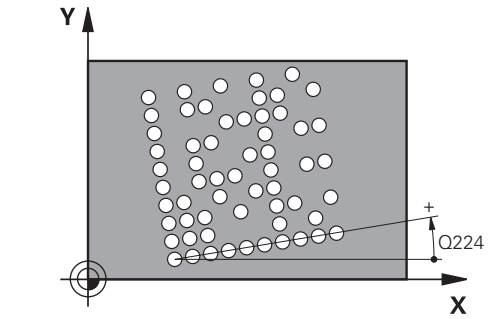
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

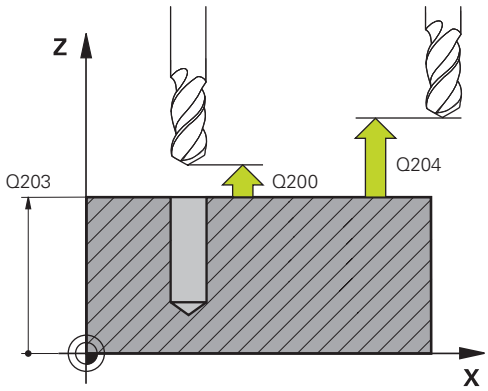
Cycle 224 is DEF-active. In addition, Cycle 224 automatically calls the last defined fixed cycle.

Cycle parameters

- **Q225 Starting point in 1st axis?** (absolute): Coordinate in the lower left corner of the code in the principal axis
- **Q226 Starting point in 2nd axis?** (absolute): Definition of a coordinate in the lower left corner of the code in the secondary axis
- **QS501 Text input?** Enter the text to be converted, in quotation marks. Maximum text length: 255 characters
- **Q458 Cell size/Pattern size(1/2)?**: Define how the DataMatrix code will be described in **Q459**:
  1: Cell spacing
  2: Pattern size
- **Q459 Size for pattern?** (incremental): Defines the cell spacing or the pattern size:
  If **Q458=1**: Spacing between the first and second cell (based on the cell centers)
  If **Q458=2**: Spacing between the first and last cell (based on the cell centers)
  Input range: 0 to 99999.9999
- **Q224 Angle of rotation?** (absolute): Angle by which the entire pattern is rotated. The center of rotation is located at the starting point
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999





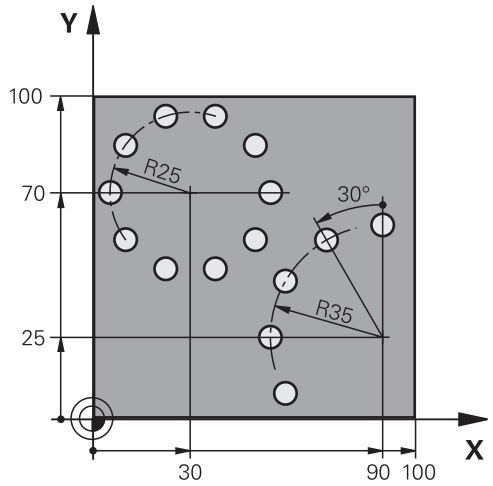


Example

```
54 CYCL DEF 224  DATAMATRIX CODE
    PATTERN
    Q225=+0      ;STARTNG PNT 1ST AXIS
    Q226=+0      ;STARTNG PNT 2ND AXIS
    QS501="ABC"  ;TEXT
    Q458=+1      ;SIZE SELECTION
    Q459=+1      ;SIZE
    Q224=+0      ;ANGLE OF ROTATION
    Q200=+2      ;SET-UP CLEARANCE
    Q203=+0      ;SURFACE COORDINATE
    Q204=50      ;2ND SET-UP CLEARANCE
```

8.5 Programming Examples

Example: Polar hole patterns



0 BEGIN PGM HOLEPAT MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call
4 L Z+250 R0 FMAX M3	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 CYCL DEF 220 POLAR PATTERN	Define cycle for polar point pattern 1, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220
Q216=+30 ;CENTER IN 1ST AXIS	
Q217=+70 ;CENTER IN 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIAMETR	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	

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

9

Fixed Cycles: Contour Pocket

9.1 SL Cycles

Fundamentals

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

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, you should always run a graphical program test before machining! This is a simple way of finding out whether the program calculated by the control will provide the desired results.

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

Characteristics of the subprograms

- Coordinate transformations are permitted—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 control recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR
- The control 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 NC 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
...
```

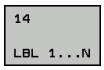
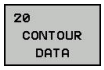
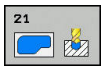
Characteristics of the fixed cycles

- The control automatically positions the tool to the set-up clearance before each cycle. You must move the tool to a safe position before the cycle call
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling

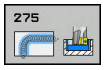
The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

```
60 LBL 0
...
99 END PGM  SL2 MM
```

Overview

Soft key	Cycle	Page
	14 CONTOUR (compulsory)	261
	20 CONTOUR DATA (compulsory)	266
	21 PILOT DRILLING (optional)	268
	22 ROUGH-OUT (compulsory)	270
	23 FLOOR FINISHING (optional)	275
	24 SIDE FINISHING (optional)	277

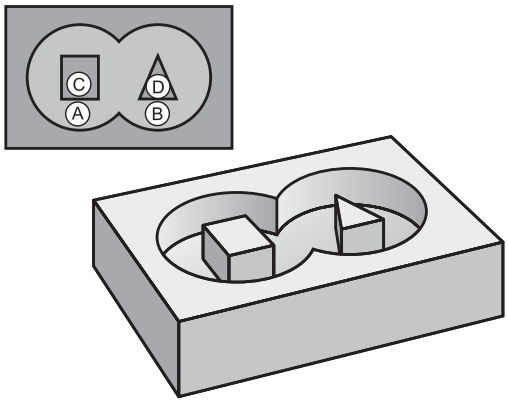
Enhanced cycles:

Soft key	Cycle	Page
	270 CONTOUR TRAIN DATA	280
	25 CONTOUR TRAIN	281
	275 TROCHOIDAL SLOT	285
	276 THREE-D CONT. TRAIN	291

9.2 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.

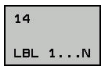


This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

Cycle 14 is DEF-active which means that it becomes effective as soon as it is defined in the NC program.

You can list up to 12 subprograms (subcontours) in Cycle 14.

Cycle parameters

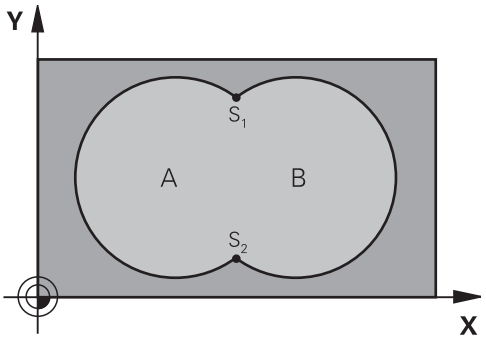


- **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm each input with the ENT key. Confirm each input with the **END** key. Input of up to 12 subprogram numbers: 1 to 65 535

9.3 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.



Example

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

Subprograms: overlapping pockets

The following examples show contour subprograms that are called by Cycle 14 CONTOUR in a main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2. They need not be programmed.

The pockets are programmed as full circles.

Subprogram 1: Pocket A

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

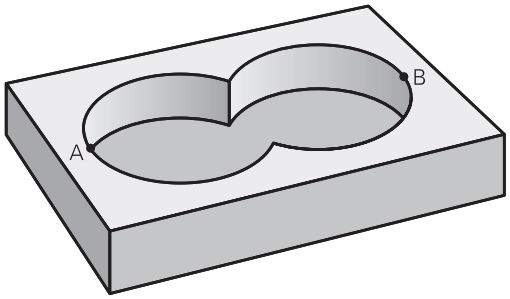
Subprogram 2: Pocket B

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

Area of inclusion

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

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



Surface A:

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

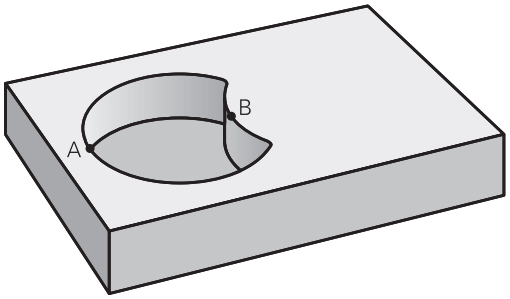
Surface B:

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

Area of exclusion

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

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



Surface A:

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

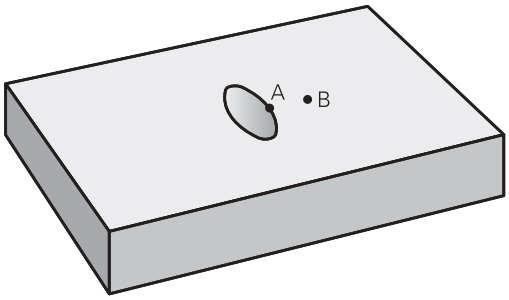
Surface B:

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

Area of intersection

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

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



Surface A:

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

Surface B:

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

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

Please note while programming:

Use Cycle 20 to program machining data for the subprograms describing the subcontours.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH = 0, the control performs the cycle at the depth 0.

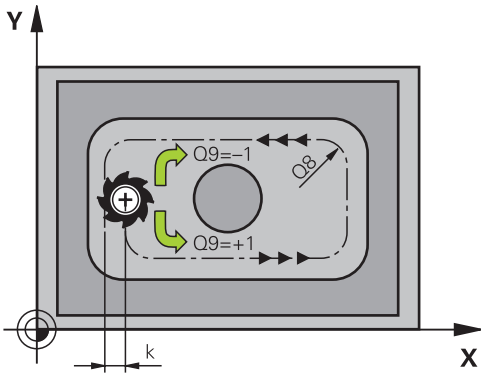
If you are using the SL cycles in **Q** parameter programs, the cycle parameters **Q1** to **Q20** cannot be used as program parameters.

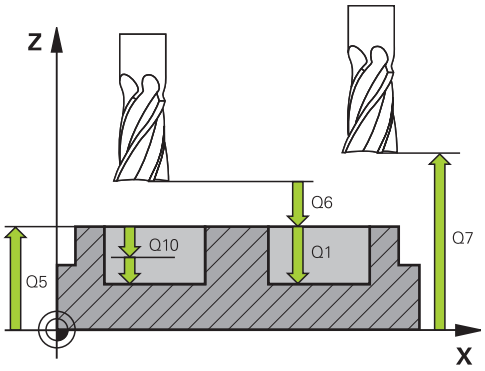
Cycle parameters

20 CONTOUR DATA

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

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





Example

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

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

Cycle run

You use Cycle 21 PILOT DRILLING if you subsequently do not use a center-cut end mill (ISO 1641) for clearing out your contour. This cycle drills a hole in the area that will be roughed out later 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** 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 tool first positions the tool in the plane (the position results from the contour that you previously defined with Cycle 14 or SEL CONTOUR and the information on the rough-out tool)
2 The tool then moves at rapid traverse **FMAX** to the set-up clearance. (Define the set-up clearance in Cycle 20 CONTOUR DATA).
3 The tool drills from the current position to the first plunging depth at the programmed feed rate **F**.
4 Then, the tool retracts at rapid traverse **FMAX** to the starting position and advances again to the first plunging depth minus the advanced stop distance t
5 The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: t = 0.6 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
6 The tool then advances with another infeed at the programmed feed rate **F**.
7 The control repeats this procedure (steps 1 to 4) until the 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 **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

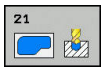
When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.

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

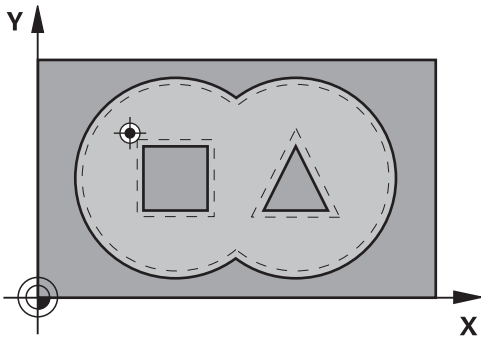
If **Q13**=0, the control uses the data of the tool that is currently in the spindle.

After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter to **ToolAxClearanceHeight**.

Cycle parameters



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



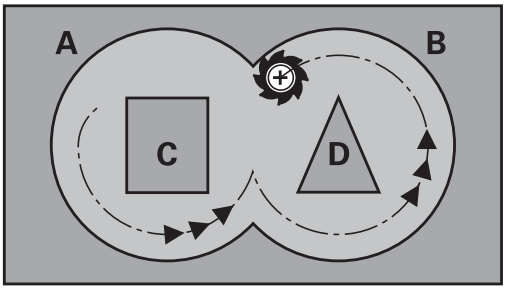
Example

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

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

Cycle run

1 The control positions the tool above the cutter infeed point, taking the allowance for side into account
2 After reaching the first plunging depth, the tool mills the contour in an outward direction at the programmed milling feed rate **Q12**
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 control 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 **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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 control 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.

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

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

Cycle parameters

- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: –99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control 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 control 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 control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered
- **Q19 Feed rate for reciprocation?**: Traversing speed of the tool in mm/min during reciprocating plunge-cut. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q12**. Input range: 0 to 99999.9999; alternatively **FMAX, FAUTO**

Example

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

- **Q401 Feed rate factor in %?**: Percentage factor by which the control reduces the machining feed rate (**Q12**) as soon as the tool moves with its entire circumference within the material 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 control then reduces the feed rate as per your definition at transitions and narrow places, reducing the total machining time. Input range: 0.0001 to 100.0000
- **Q404 Fine roughing strategy (0/1)?**: Specify the fine roughing behavior of the control if the radius of the fine roughing tool is greater than or equal to half the radius of the coarse roughing tool:
  Q404=0:
  The control moves the tool between the areas to be fine-roughed at the current depth along the contour
  Q404=1:
  The control 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

9.7 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 smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control 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 or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary
- Cycle 22 ROUGHING, if necessary

Cycle run

1 The control 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 smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control 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 **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

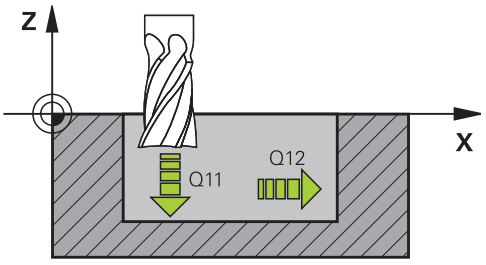
The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

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.

Cycle parameters

- **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q12**. Input range: 0 to 99999.9999; alternatively **FMAX, FAUTO**



Example

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

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

Cycle run

1. The control 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 control moves the tool when approaching the contour
2. The control then moves the tool to the first plunging depth using the feed rate for plunging
3. The contour is approached on a tangential arc and machined up to the end. Each subcontour is finished separately
4. The tool moves on a tangential helical arc when approaching the finishing contour or retracting from it. The starting height of the helix is 1/25 of the set-up clearance **Q6**, but max. the remaining last plunging depth above the final depth
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 **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The sum of finishing allowance for the 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 finishing allowance for the 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.

Cycle 24 can also be used for contour milling. In that case, you must do the following:

- define the contour to be milled as a single island (without pocket limit)
- 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 control 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.

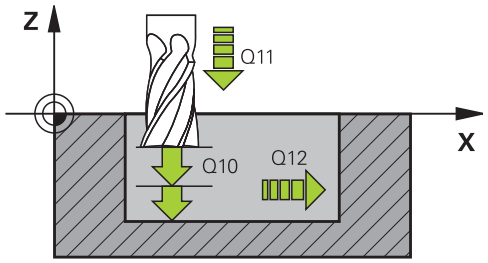
The starting point calculated by the control also depends on the machining sequence. If you select the finishing cycle with the GOTO key and then start the NC program, the starting point can be at a different location from where it would be if you execute the NC program in the defined sequence.

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

You can execute this cycle using a grinding tool.

Cycle parameters

- **Q9 Direction of rotation? cw = -1**: Machining direction:
  +1: Counterclockwise rotation
  –1: Clockwise rotation
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: –99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q14 Finishing allowance for side?** (incremental): The finishing allowance for the 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
- **Q438 Number/name of rough-out tool? Q438** or **QS438**: Number or name of the tool with which the control roughed out the contour pocket. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. Input range if a number is entered: –1 to +32767.9
  Q438=–1: The control assumes that the tool last used was the rough-out tool (default behavior)
  Q438=0: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0.



Example

61 CYCL DEF 24 SIDE FINISHING	
Q9=+1	;ROTATIONAL DIRECTION
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q14=+0	;ALLOWANCE FOR SIDE
Q438=-1	;NUMBER/NAME OF ROUGH-OUT TOOL?

9.9 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

Define Cycle 270 before Cycle 25.

Cycle parameters

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

Example

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

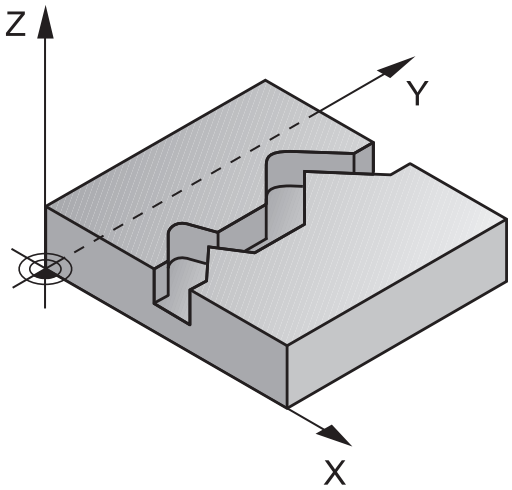
9.10 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125)

Cycle run

In conjunction with Cycle 14 CONTOUR, 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 control monitors the operation to prevent undercuts and contour damage (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
- Machining can done throughout by up-cut or by climb milling. The type of milling will even be retained if the contours were 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!

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

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

If 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.

You can execute this cycle using a grinding tool.

Cycle parameters

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

Example

```
62 CYCL DEF 25  CONTOUR TRAIN
    Q1=-20      ;MILLING DEPTH
    Q3=+0       ;ALLOWANCE FOR SIDE
    Q5=+0       ;SURFACE COORDINATE
    Q7=+50      ;CLEARANCE HEIGHT
    Q10=+5      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLNGNG
    Q12=350     ;FEED RATE F. ROUGHNG
    Q15=-1      ;CLIMB OR UP-CUT
    Q18=0       ;COARSE ROUGHING
                 TOOL
    Q446=+0.01;RESIDUAL MATERIAL
    Q447=+10    ;CONNECTION DISTANCE
    Q448=+2     ;PATH EXTENSION
```

- **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control 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 control 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 control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered
- **Q446 Accepted residual material?** Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input range: 0.001 to 9.999
- **Q447 Maximum connection distance?** Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input range: 0 to 999.9999
- **Q448 Path extension?** Length by which the tool path is extended at the beginning and end of the end of a contour area. The control always extends the tool path parallel to the contour. Input range: 0 to 99.999

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

Cycle run

In conjunction with Cycle 14 **CONTOUR**, 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 can be combined as the equally distributed cutting forces prevent increased wear of the tool. When indexable 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** (option 45).

Further information: User's Manual for 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

In case of a closed slot, the contour description must always start with an straight line block (**L** block).

1. Following the positioning logic, the tool moves to the starting point of the contour description and moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2. The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the 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 control moves the tool to clearance height and returns it 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. If a finishing allowance has been defined, the control finishes the slot walls, in multiple infeeds, if so specified. Starting from the defined starting point, the control approaches the slot wall tangentially. Climb or up-cut milling is taken into consideration.

Program structure: Machining with SL cycles

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

Roughing with 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 control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the 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 control moves the tool to clearance height and returns it 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 If a finishing allowance has been defined, the control finishes the slot walls (in multiple infeeds if specified). The control approaches the slot wall starting from the defined starting point of the **APPR** block. Climb or up-cut milling is taken into consideration

Please note while programming:

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The algebraic sign for the DEPTH cycle parameter 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.

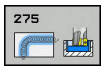
Define the center line of the slot with all available path functions in the contour subprogram.

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

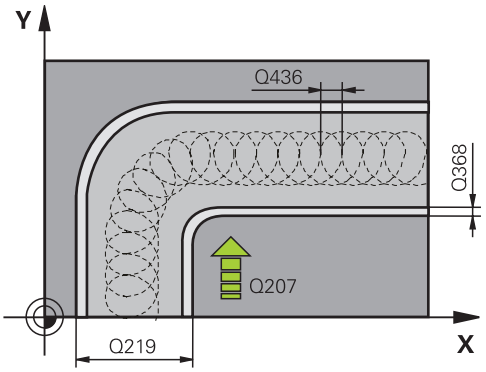
In conjunction with Cycle 275, the control does not require Cycle 20 CONTOUR DATA.

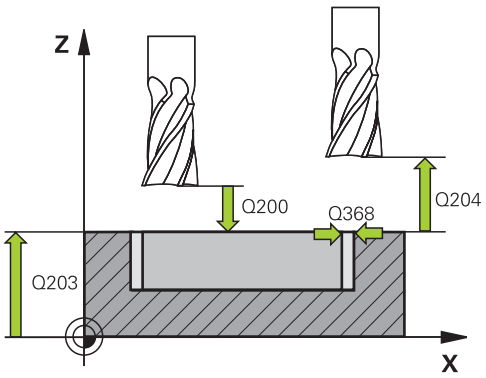
The starting point of a closed slot must not be located in a contour corner.

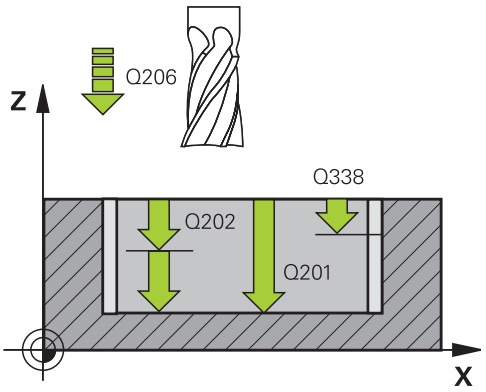
Cycle parameters



- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q219 Width of slot?** (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q436 Feed per revolution?** (absolute): Value by which the control moves the tool in the machining direction per revolution. Input range: 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: –99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999







- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999
- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  0 = vertical plunging. The control plunges the tool 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. Otherwise, the control will display an error message
  Alternatively: **PREDEF**

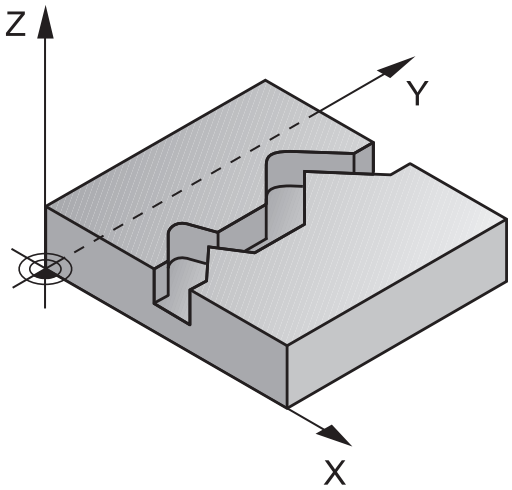
Example

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

- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
  0: Feed rate with respect to the tool center point path
  1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
  2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
  3: Feed rate always refers to the cutting edge

9.12 THREE-D CONT. TRAIN (Cycle 276, DIN/ISO: G276)

Cycle run



In conjunction with Cycle 14 CONTOUR and Cycle 270 **CONTOUR TRAIN DATA**, this cycle facilitates the machining of open and closed contours. You can also work with automatic residual material detection. This way you can subsequently complete e.g. inside corners with a smaller tool.

In contrast to Cycle 25 **CONTOUR TRAIN**, Cycle 276 **THREE-D CONT. TRAIN** also processes tool axis coordinates defined in the contour subprogram. This cycle can thus machine three-dimensional contours.

We recommend that you program Cycle 270 **CONTOUR TRAIN DATA** before Cycle 276 **THREE-D CONT. TRAIN**.

Machining a contour without infeed: Milling depth **Q1**=0

1 The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle 270 **CONTOUR TRAIN DATA**, e.g. in Type of approach. The control then moves the tool to the first plunging depth
2 According to the previously defined Cycle 270 **CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3 At the end of the contour, the tool will be retracted as defined in Cycle 270 **CONTOUR TRAIN DATA**
4 Finally, the control retracts the tool to the clearance height.

Machining a contour with infeed: Milling depth **Q1** not equal to 0 and plunging depth **Q10** are defined

1 The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle 270 **CONTOUR TRAIN DATA**, e.g. in Type of approach. The control then moves the tool to the first plunging depth
2 According to the previously defined Cycle 270 **CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3 If you selected machining with Climb milling and up-cut milling (**Q15**=0), the control will perform a reciprocation movement. The infeed movement (plunging) will be performed at the end and at the starting point of the contour. If **Q15** is not equal to 0, the tool is moved to clearance height and returned to the starting point of machining. From there, the control moves the tool to the next plunging depth
4 The departure will be performed as defined in Cycle 270 **CONTOUR TRAIN DATA**
5 This process is repeated until the programmed depth is reached
6 Finally, the control retracts the tool to the clearance height.

Please note while programming:

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

NOTICE

Danger of collision!

A collision may occur if you position the tool behind an obstacle before the cycle is called.

- Before the cycle call, position the tool in such a way that the tool can approach the starting point of the contour without collision
- If the position of the tool is below the clearance height when the cycle is called, the control will issue an error message

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The first NC block in the contour subprogram must contain values in all of the three axes X, Y and Z.

If you program **APPR** and **DEP** blocks for contour approach and departure, the control monitors whether the execution of any of these blocks would damage the contour

The algebraic sign for the depth parameter determines the working direction. If you program DEPTH=0, the cycle will use the tool axis coordinates that have been specified in the contour subprogram.

If you are using Cycle 25 CONTOUR TRAIN, you can define only one subprogram in the CONTOUR cycle.

We recommend that you use Cycle 270 CONTOUR TRAIN DATA in conjunction with Cycle 276. Cycle 20 CONTOUR DATA, however, is not required.

If 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.

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

Cycle parameters

- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: –99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: –99999.9999 to 99999.9999
- **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: –99999.9999 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: –99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q15 Climb or up-cut? up-cut = -1**:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  Climb milling and up-cut milling alternately in several infeeds: Input value = 0
- **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control 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 control 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 control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered

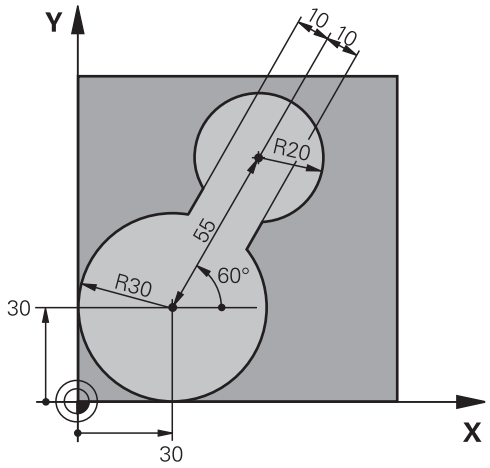
Example

```
62 CYCL DEF 276  THREE-D CONT.
   TRAIN
   Q1=-20        ;MILLING DEPTH
   Q3=+0         ;ALLOWANCE FOR SIDE
   Q7=+50        ;CLEARANCE HEIGHT
   Q10=-5        ;PLUNGING DEPTH
   Q11=150       ;FEED RATE FOR PLNGNG
   Q12=500       ;FEED RATE F. ROUGHNG
   Q15=+1        ;CLIMB OR UP-CUT
   Q18=0         ;COARSE ROUGHING
                 TOOL
   Q446=+0.01;RESIDUAL MATERIAL
   Q447=+10      ;CONNECTION DISTANCE
   Q448=+2       ;PATH EXTENSION
```

- **Q446 Accepted residual material?** Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input range: 0.001 to 9.999
- **Q447 Maximum connection distance?** Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input range: 0 to 999.9999
- **Q448 Path extension?** Length by which the tool path is extended at the beginning and end of the end of a contour area. The control always extends the tool path parallel to the contour. Input range: 0 to 99.999

9.13 Programming Examples

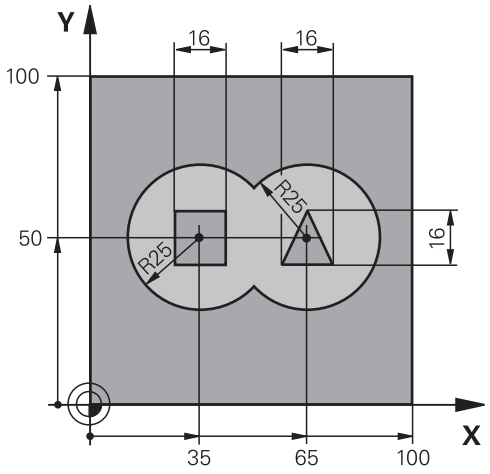
Example: Roughing-out and fine-roughing a pocket



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

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

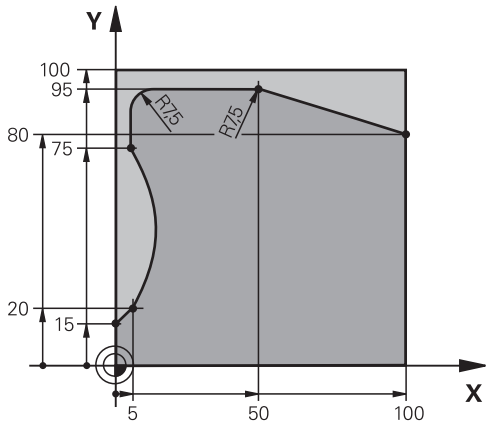
Example: Pilot drilling, roughing-out and finishing overlapping contours



0 BEGIN PGM C21 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: drill, diameter 12
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 21 PILOT DRILLING	Cycle definition: pilot drilling
Q10=5 ;PLUNGING DEPTH	
Q11=250 ;FEED RATE FOR PLNGNG	
Q13=2 ;ROUGH-OUT TOOL	
9 CYCL CALL M3	Cycle call: pilot drilling
10 L +250 R0 FMAX M6	Retract the tool
11 TOOL CALL 2 Z S3000	Tool call: roughing/finishing, diameter 12
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	

Q12=350 ;FEED RATE F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=30000 ;RETRACTION FEED RATE	
13 CYCL CALL M3	Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING	Cycle definition: floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE F. ROUGHNG	
Q208=30000 ;RETRACTION FEED RATE	
15 CYCL CALL	Cycle call: floor finishing
16 CYCL DEF 24 SIDE FINISHING	Cycle definition: side finishing
Q9=+1 ;ROTATIONAL DIRECTION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE F. ROUGHNG	
Q14=+0 ;ALLOWANCE FOR SIDE	
17 CYCL CALL	Cycle call: side finishing
18 L Z+250 R0 FMAX M2	Retract the tool, end program
19 LBL 1	Contour subprogram 1: left pocket
20 CC X+35 Y+50	
21 L X+10 Y+50 RR	
22 C X+10 DR-	
23 LBL 0	
24 LBL 2	Contour subprogram 2: right pocket
25 CC X+65 Y+50	
26 L X+90 Y+50 RR	
27 C X+90 DR-	
28 LBL 0	
29 LBL 3	Contour subprogram 3: left square 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: right triangular island
37 L X+65 Y+42 RL	
38 L X+57	
39 L X+65 Y+58	
40 L X+73 Y+42	
41 LBL 0	
42 END PGM C21 MM	

Example: Contour train



0 BEGIN PGM C25 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2000	Tool call, diameter 20
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 25 CONTOUR TRAIN	Define machining parameters
Q1=-20 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q5=+0 ;SURFACE COORDINATE	
Q7=+250 ;CLEARANCE HEIGHT	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE F. ROUGHNG	
Q15=+1 ;CLIMB OR UP-CUT	
Q466= 0.01 ;RESIDUAL MATERIAL	
Q447=+10 ;CONNECTION DISTANCE	
Q448=+2 ;PATH EXTENSION	
8 CYCL CALL M3	Cycle call
9 L Z+250 R0 FMAX M2	Retract the tool, end program
10 LBL 1	Contour subprogram
11 L X+0 Y+15 RL	
12 L X+5 Y+20	
13 CT X+5 Y+75	
14 L Y+95	
15 RND R7.5	
16 L X+50	
17 RND R7.5	

```
18 L  X+100 Y+80
19 LBL 0
20 END PGM  C25 MM
```

10

Machining Cycles: Optimized contour milling

10.1 OCM cycles (option 167)

OCM fundamentals

General information

Using OCM cycles (**Optimized Contour Milling**), you can combine subcontours to form complex contours. These cycles are more powerful than Cycles 22 to 24. OCM cycles provide the following additional functions:

- When roughing, the control will maintain the specified tool angle precisely
- Besides pockets, you can also machine islands and open pockets

You can program up to 16 384 contour elements in one OCM cycle.

OCM cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, you should always run a graphical program test before machining! This is a simple way of finding out whether the program calculated by the control will provide the desired results.

Contact angle

When roughing, the control will retain the tool angle precisely. The tool angle can be defined implicitly by specifying an overlap factor. The maximum overlap factor is 1; this corresponds to an angle of 90° max.

Contour

Specify the contour with **CONTOUR DEF**. The first contour may be a pocket or a boundary. The next contours can be programmed as islands or pockets.

To program open pockets, use a boundary and an island.

Proceed as follows:

- Program **CONTOUR DEF**
- Define the first contour as a pocket and the second one as an island
- Define the **OCM CONTOUR DATA** cycle
- In cycle parameter **Q569**, program the value 1

> The control will interpret the first contour as an open boundary instead of a pocket. Thus, the open boundary and the island programmed subsequently are combined to form an open pocket.

An example is provided after the description of the OCM cycles, see "Example: Open pocket and fine roughing with OCM cycles", Page 316

> Subsequently defined contours that are outside the first contour will not be considered.

Closed pockets can also be defined in Cycle 14.

The machining dimensions, such as milling depth, finishing allowances and clearance height, can be entered in Cycle 271 as **OCM CONTOUR DATA**.

Machining operation

When roughing, these cycles allow you to use larger tools for the first roughing passes and then smaller tools to take off the residual material. For finishing, the material roughed out will be taken into consideration.

Example

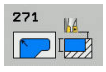
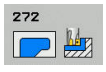
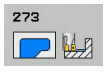
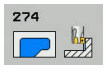
You defined a Ø20 mm rough-out tool. For roughing, this results in minimum inside radii of 10 mm (cycle parameter **Q578** Radius factor on inside corners will not be taken into account in this example). In the next step, you will finish the contour. For this purpose, define a Ø10 mm finishing cutter. In this case, the maximum inside radii would be 5 mm. Finishing cycles will also consider the previous machining steps, depending on **Q438**, so that the smallest finishing inside radii will be 10 mm. Thus, the finishing cutter will be protected from overload.

Program structure: Machining with OCM cycles

```
0 BEGIN PGM  SL2 MM
...
12 CONTOUR DEF ...
13 CYCL DEF 271 OCM CONTOUR DATA ...
...
16 CYCL DEF 272  OCM ROUGHING ...
17 CYCL CALL
...
18 CYCL DEF 273 OCM FINISHING FLOOR  ...
19 CYCL CALL
...
22 CYCL DEF 274  OCM FINISHING SIDE ...
23 CYCL CALL
50 L  Z+250 R0  FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM  SL2 MM
```

Overview

OCM cycles:

Soft key	Cycle	Page
	271 OCM CONTOUR DATA	307
	272 OCM ROUGHING	309
	273 OCM FINISHING FLOOR	312
	274 OCM FINISHING SIDE	314

10.2 OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167)

Cycle run

Use Cycle 271 **OCM CONTOUR DATA** to program machining data for the contour or the subprograms describing the subcontours. In addition, Cycle 271 allows you to define an open boundary for a pocket.

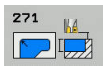
Please note while programming!

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

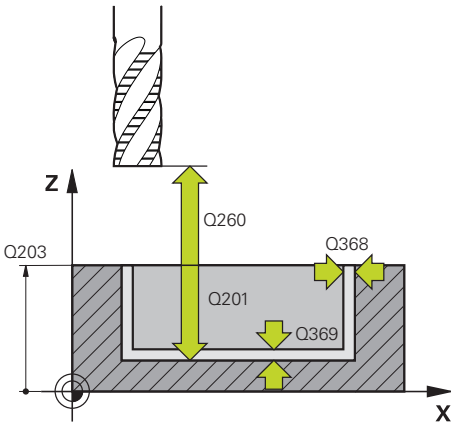
Cycle 271 is DEF-active, which means that it becomes active as soon as it is defined in the NC program.

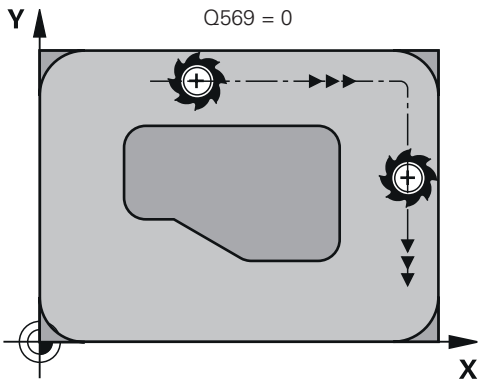
The machining data entered in Cycle 271 are valid for Cycles 272 to 274.

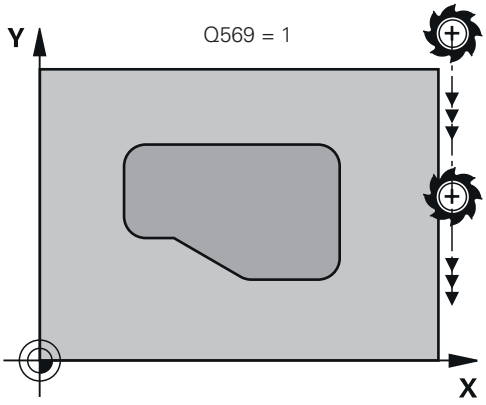
Cycle parameters



- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: –99999.9999 to 0
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: –99999.9999 to 99999.9999
- **Q578 Radius factor on inside corners?** The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**. Input range: 0.05 to 0.99
- **Q569 Is the first pocket a boundary?** Define the boundary:
  0: The first contour in CONTOUR DEF is interpreted as a pocket.
  1: The first contour in CONTOUR DEF is interpreted as an open boundary.







Example

```
59 CYCL DEF 271  OCM CONTOUR DATA
   Q203=+0      ;SURFACE COORDINATE
   Q201=-20     ;DEPTH
   Q368=+0      ;ALLOWANCE FOR SIDE
   Q369=+0      ;ALLOWANCE FOR FLOOR
   Q260=+100    ;CLEARANCE HEIGHT
   Q578=+0.2    ;INSIDE CORNER FACTOR
   Q569=+0      ;OPEN BOUNDARY
```

10.3 OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167)

Cycle run

Use Cycle 272 **OCM ROUGHING** to define the technology data for roughing.

Before calling Cycle 272 you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle 14 **CONTOUR GEOMETRY**
- Cycle 271 **OCM CONTOUR DATA**

1 The tool uses positioning logic to move to the starting point.

2 The control determines the starting point automatically based on the pre-positioning and the programmed contour.
   - If you program **Q569**=0, the tool plunges into the material in a helical movement to reach the first plunging depth. The finishing allowance for the side is taken into account
   - If you program **Q569**=1, the tool plunges outside the open boundary

3 After reaching the first plunging depth, the tool mills the contour in an outward or inward direction (depending on **Q569**) at the programmed milling feed rate **Q207**

4 In the next step, the control moves the tool to the next plunging depth and repeats the roughing procedure until the programmed depth is reached

5 Finally, the tool retracts in the tool axis to the clearance height.

Please note while programming!

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

CONTOUR DEF will reset the tool radius that was used last. If you run this machining cycle with Q438=–1 after CONTOUR DEF, the control assumes that no pre-machining has taken place yet.

If required, use a center-cut end mill (ISO 1641).

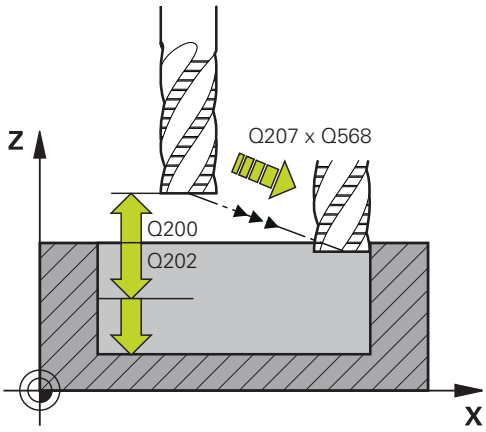
If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.

Define the plunging behavior of Cycle 272 in the **ANGLE** and **LCUTS** columns of the tool table:

- If **ANGLE** is between 0.1° and 89.999° in the tool table, the control plunges helically using the defined **ANGLE**
- If **ANGLE** is less than 0.1° or larger than 90° in the tool table, the control displays an error message
- If the geometry of the workpiece (slot) prevents helical plunging, the control displays a note that plunging is not possible at this position. You can then use a smaller tool for re-machining.

Cycle parameters

- **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999
- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.01 to 1; alternatively **PREDEF**
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q568 Factor for plunging feed rate?** Factor by which the control reduces the feed rate **Q207** for downfeed into the material. Input range: 0.1 to 1
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. In mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q438 Number/name of rough-out tool? Q438** or **QS438**: Number or name of the tool with which the control roughed out the contour pocket. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. Input range if a number is entered: –1 to +32767.9
  Q438=–1: The control assumes that the tool last used in Cycle 272 was the rough-out tool (default behavior)
  Q438=0: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0.



Example

```
59 CYCL DEF 272  OCM ROUGHING
    Q202=+5      ;PLUNGING DEPTH
    Q370=+0.4    ;TOOL PATH OVERLAP
    Q207=+500    ;FEED RATE MILLING
    Q568=+0.6    ;PLUNGING FACTOR
    Q253=+750    ;F PRE-POSITIONING
    Q200=+2      ;SAFETY CLEARANCE
    Q438=-1      ;ROUGH-OUT TOOL
    Q577=+0.2    ;APPROACH RADIUS
                 FACTOR
    Q351=+1      ;CLIMB OR UP-CUT
```

- **Q577 Factor for appr./dept. radius?** Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input range: 0.15 to 0.99
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

10.4 OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167)

Cycle run

With Cycle 273 **OCM FINISHING FLOOR**, you can clear the finishing allowance for side that is programmed in Cycle 271.

Before calling Cycle 273 you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle 14 **CONTOUR GEOMETRY**
- Cycle 271 **OCM CONTOUR DATA**
- Cycle 272 **OCM ROUGHING**, if applicable**OCM ROUGHING**

1 The control positions the tool to the clearance height at rapid traverse **FMAX**

2 The tool then moves in the tool axis at the feed rate **Q385**

3 The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control 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

Please note while programming!

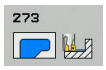
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.

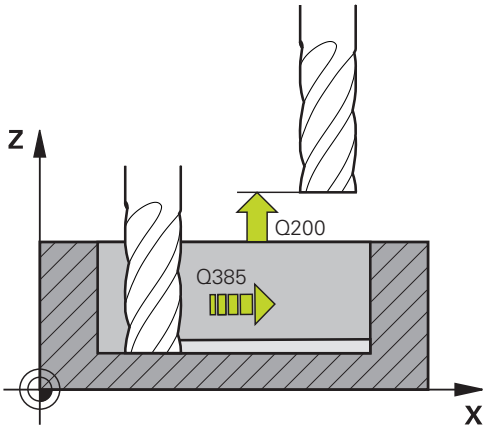
For finishing with Cycle 273, the tool always works in climb milling mode.

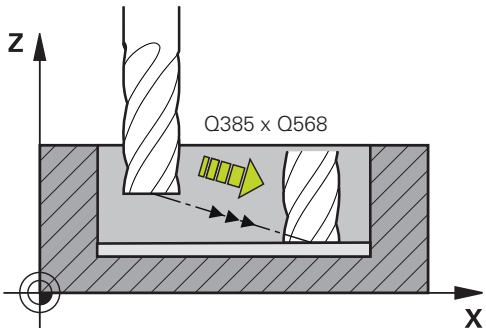
Specify a rough-out tool in cycle parameter **Q438**, otherwise, the control will generate an error message.

Cycle parameters



- **Q370 Path overlap factor?**: **Q370** x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.0001 to 1.9999; alternatively **PREDEF**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q568 Factor for plunging feed rate?** Factor by which the control reduces the feed rate **Q385** for downfeed into the material. Input range: 0.1 to 1
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. In mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q438 Number/name of rough-out tool? Q438** or **QS438**: Number or name of the tool with which the control roughed out the contour pocket. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. Input range if a number is entered: –1 to +32767.9
  Q438=–1: The control assumes that the tool last used was the rough-out tool (default behavior)





Example

```
60 CYCL DEF 273  OCM FINISHING
   FLOOR
   Q370=+1     ;TOOL PATH OVERLAP
   Q385=+500   ;FINISHING FEED RATE
   Q568=+0.3   ;PLUNGING FACTOR
   Q253=+750   ;F PRE-POSITIONING
   Q200=+2     ;SET-UP CLEARANCE
   Q438=-1     ;ROUGH-OUT TOOL
```

10.5 OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167)

Cycle run

With Cycle 274 **OCM FINISHING SIDE**, you can clear the finishing allowance for the side that is programmed in Cycle 271. You can run this cycle in climb or up-cut milling.

Before calling Cycle 274 you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle 14 **CONTOUR GEOMETRY**
- Cycle 271 **OCM CONTOUR DATA**
- Cycle 272 **OCM ROUGHING**, if applicable
- Cycle 273 **OCM FINISHING FLOOR**, if applicable

1 The control 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 control moves the tool when approaching the contour

2 The control then moves the tool to the first plunging depth using the feed rate for plunging

3 The tool approaches and moves along the contour helically on a tangential arc until the entire contour is finished. Each subcontour is finished separately

4 Finally, the tool retracts in the tool axis to the clearance height.

Cycle 274 can also be used for contour milling.

Proceed as follows:

- Define the contour to be milled as a single island (without pocket boundary)
- Enter the finishing allowance (**Q368**) in Cycle 271 to be greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Please note while programming!

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

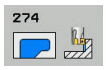
The finishing allowance for the side **Q14** is left over after finishing. It must be smaller than the allowance in Cycle 271.

The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour and the allowance programmed in Cycle 271.

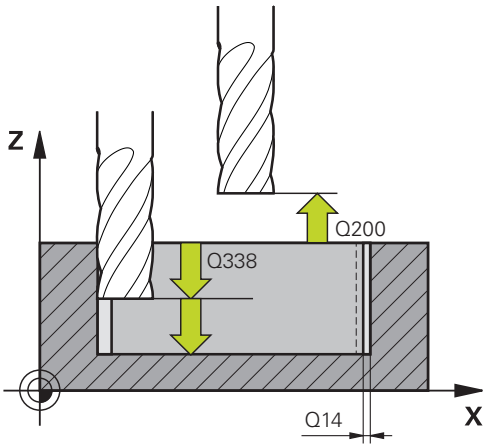
Specify a rough-out tool in cycle parameter **Q438**, otherwise, the control will generate an error message.

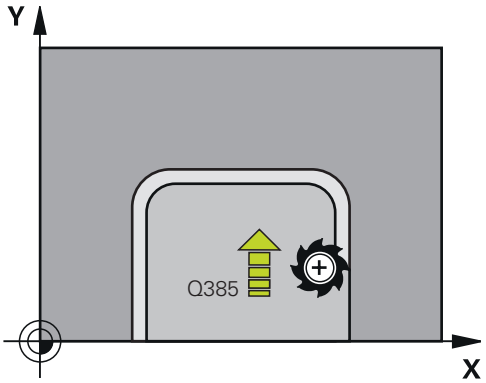
You can execute this cycle using a grinding tool.

Cycle parameters



- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. In mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q14 Finishing allowance for side?** (incremental): The finishing allowance for the side **Q14** is left over after finishing. (This allowance must be smaller than the allowance in Cycle 271.) Input range: –99999.9999 to 99999.9999
- **Q438 Number/name of rough-out tool? Q438** or **QS438**: Number or name of the tool with which the control roughed out the contour pocket. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. Input range if a number is entered: –1 to +32767.9
  Q438=–1: The control assumes that the tool last used was the rough-out tool (default behavior)
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)





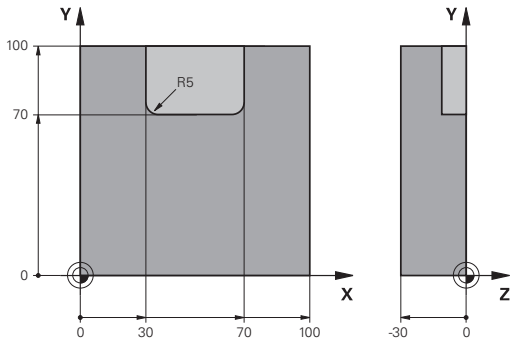
Example

```
61 CYCL DEF 274  OCM FINISHING SIDE
   Q338=+0      ;INFEED FOR FINISHING
   Q385=+500    ;FINISHING FEED RATE
   Q253=+750    ;F PRE-POSITIONING
   Q200=+2      ;SET-UP CLEARANCE
   Q14=+0       ;ALLOWANCE FOR SIDE
   Q438=-1      ;NUMBER/NAME OF
                 ROUGH-OUT TOOL?
   Q351=+1      ;CLIMB OR UP-CUT
```

10.6 Programming examples

Example: Open pocket and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will program an open pocket. For this purpose, use a boundary and an island.



Program sequence

- Tool call: roughing cutter
- Program **CONTOUR DEF**
- Define Cycle 271
- Define and call Cycle 272
- Tool call: finishing cutter
- Define and call Cycle 273
- Define and call Cycle 274

0 BEGIN PGM OCM_POCKET MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL "MILL_D20" Z S8000 F1500	Tool call, diameter 20
4 M3	
5 L Z+250 R0 FMAX	
6 L X+0 Y+0 R0 FMAX	
7 CONTOUR DEF P1 = LBL 1 I2 = LBL 2	
8 CYCL DEF 271 OCM CONTOUR DATA	Define machining parameters
Q203=+0 ;SURFACE COORDINATE	
Q201=-10 ;DEPTH	
Q368=+0.5 ;ALLOWANCE FOR SIDE	
Q369=+0.5 ;ALLOWANCE FOR FLOOR	
Q260=+100 ;SICHERE HOEHE	
Q578=+0.2 ;INSIDE CORNER FACTOR	
Q569=+1 ;OPEN BOUNDARY	
9 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+5 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= AUTO ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=+0 ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
10 CYCL CALL	Cycle call
11 TOOL CALL "MILL_D8" Z S8000 F1500	Tool call, diameter 8

12 M3	
13 L Z+250 R0 FMAX	
14 L X+0 Y+0 R0 FMAX	
15 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+5 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= AUTO ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
QS438="MILL_D20" ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
16 CYCL CALL	Cycle call
17 TOOL CALL "MILL_D6_FINISH" Z S10000 F2000	Tool call, diameter 6
18 M3	
19 L Z+250 R0 FMAX	
20 L X+0 Y+0 R0 FMAX	
21 CYCL DEF 273 OCM FINISHING FLOOR	Define the floor finishing cycle
Q370=+0.8 ;TOOL PATH OVERLAP	
Q385= AUTO ;FINISHING FEED RATE	
Q568=+0.3 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=-1 ;ROUGH-OUT TOOL	
22 CYCL CALL	Cycle call
23 CYCL DEF 274 OCM FINISHING SIDE	Define the side finishing cycle
Q338=+0 ;INFEED FOR FINISHING	
Q385= AUTO ;FINISHING FEED RATE	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q14=+0 ;ALLOWANCE FOR SIDE	
QS438=-1 ;ROUGH-OUT TOOL	
Q351=+1 ;CLIMB OR UP-CUT	
24 CYCL CALL	Cycle call
25 M30	End of program
26 LBL 1	Contour subprogram 1
27 L X+0 Y+0	
28 L X+100	
29 L Y+100	
30 L X+0	
31 L Y+0	
32 LBL 0	

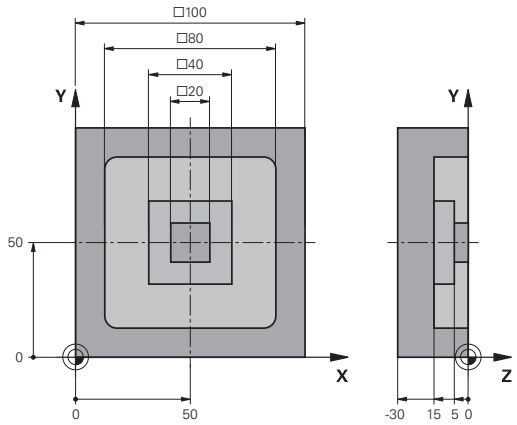
33 LBL 2	Contour subprogram 2
34 L X+0 Y+0	
35 L X+100	
36 L Y+100	
37 L X+70	
38 L Y+70	
39 RND R5	
40 L X+30	
41 L Y+100	
42 RND R5	
43 L X+0	
44 L Y+0	
45 LBL 0	
46 END PGM OCM_POCKET MM	

Example: Program various depths with OCM cycles

The following NC program illustrates the use of OCM cycles.
Define a pocket and two islands at different heights.

Program sequence

- Tool call: roughing cutter
- Program **CONTOUR DEF**
- Define Cycle 271
- Define and call Cycle 272
- Tool call: finishing cutter
- Define and call Cycle 273
- Define and call Cycle 274



0 BEGIN PGM OCM_DEPTH MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-30	Workpiece blank definition
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL "MILL_D10" Z S8000 F1500	Tool call, diameter D10
4 L Z+250 R0 FMAX M3	
5 L X+0 Y+0 R0 FMAX	
6 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 I3 = LBL 3 DEPTH5	
7 CYCL DEF 271 OCM CONTOUR DATA	Define machining parameters
Q203=+0 ;SURFACE COORDINATE	
Q201=-15 ;DEPTH	
Q368=+0.5 ;ALLOWANCE FOR SIDE	
Q369=+0.5 ;ALLOWANCE FOR FLOOR	
Q260=+100 ;SICHERE HOEHE	
Q578=+0.2 ;INSIDE CORNER FACTOR	
Q569=+0 ;OPEN BOUNDARY	
8 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+5 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= AUTO ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=+0 ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
9 CYCL CALL	Cycle call
10 TOOL CALL "MILL_D6_FINISH" Z S10000 F2000	Tool call, diameter D6
11 M3	
12 L Z+250 R0 FMAX	
13 L X+0 Y+0 R0 FMAX	

14 CYCL DEF 273 OCM FINISHING FLOOR	Define the floor finishing cycle
Q370=+0.8 ;TOOL PATH OVERLAP	
Q385= AUTO ;FINISHING FEED RATE	
Q568=+0.3 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=-1 ;ROUGH-OUT TOOL	
15 CYCL CALL	Cycle call
16 CYCL DEF 274 OCM FINISHING SIDE	Define the side finishing cycle
Q338=+0 ;INFEED FOR FINISHING	
Q385= AUTO ;FINISHING FEED RATE	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q14=+0 ;ALLOWANCE FOR SIDE	
QS438="MILL_D10" ;ROUGH-OUT TOOL	
Q351=+1 ;CLIMB OR UP-CUT	
17 CYCL CALL	Cycle call
18 M30	End of program
19 LBL 1	Contour subprogram 1
20 L X-40 Y-40	
21 L X+40	
22 L Y+40	
23 L X-40	
24 L Y-40	
25 LBL 0	
26 LBL 2	Contour subprogram 2
27 L X-10 Y-10	
28 L X+10	
29 L Y+10	
30 L X-10	
31 L Y-10	
32 LBL 0	
33 LBL 3	Contour subprogram 3
34 L X-20 Y-20	
35 L Y+20	
36 L X+20	
37 L Y-20	
38 L X-20	
39 LBL 0	
40 END PGM OCM_DEPTH MM	

11

Fixed Cycles: Cylindrical Surface

11.1 Fundamentals

Overview of cylindrical surface cycles

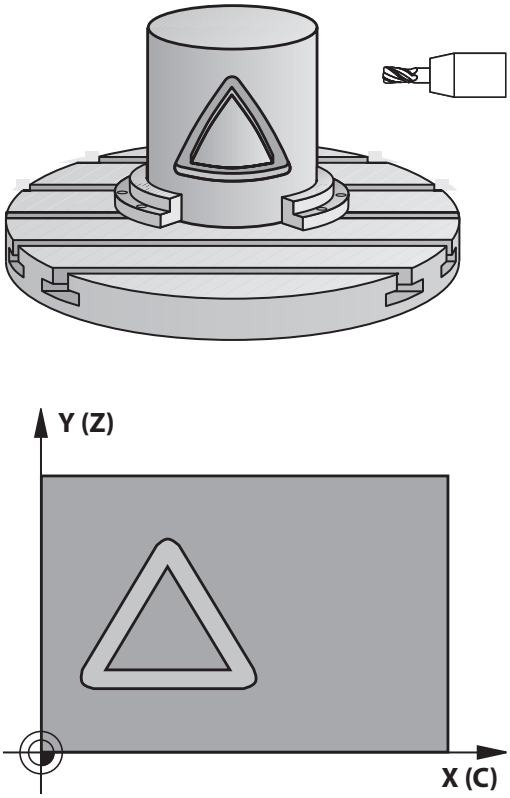
Soft key	Cycle	Page
	27 CYLINDER SURFACE	323
	28 CYLINDER SURFACE Slot milling	326
	29 CYLINDER SURFACE Ridge milling	330
	39 CYLINDER SURFACE Contour	333

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

Cycle run

Refer to your machine manual!

The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



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.

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 in the rotary axis (X coordinates) can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition using **Q17**.

1 The control positions the tool above 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 control returns the tool to the set-up clearance and returns to the infeed point
4 Steps 1 to 3 are repeated until the programmed milling depth **Q1** is reached.
5 Subsequently, the tool retracts in the tool axis to the clearance height.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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 DEPTH cycle parameter 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 preset 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 control 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.

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

Cycle parameters

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

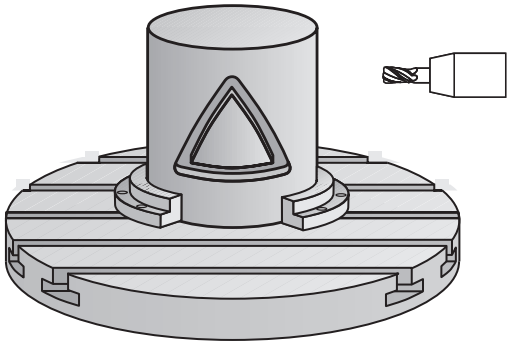
Example

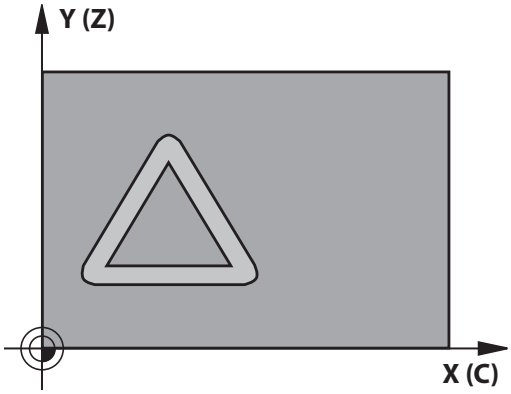
```
63 CYCL DEF 27  CYLINDER SURFACE
    Q1=-8       ;MILLING DEPTH
    Q3=+0       ;ALLOWANCE FOR SIDE
    Q6=+0       ;SET-UP CLEARANCE
    Q10=+3      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLNGNG
    Q12=350     ;FEED RATE F. ROUGHNG
    Q16=25      ;RADIUS
    Q17=0       ;TYPE OF DIMENSION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

11.3 CYLINDER SURFACE slot milling (Cycle 28, DIN/ISO: G128, option 1)

Cycle run

Refer to your machine manual!
The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.





With this cycle you can program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle 27, with this cycle, the control adjusts the tool in such a way 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 control machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.

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

1 The control positions the tool above the infeed point.
2 The control moves the tool vertically 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 **ConfigDatum CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004) parameter
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 control moves the tool to the opposite slot 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 defined the tolerance in **Q21**, the control then re-machines the slot walls to be as parallel as possible
7 Finally, the tool retracts in the tool axis to the clearance height.

Please note while programming:

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** parameter (no. 201002), to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

NOTICE

Danger of collision!

At the end, the control returns the tool 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 need not be the same as the starting position.

- Control the traversing movements of the machine
- In the simulation, control the end position of the tool after the cycle
- After the cycle, program the absolute (not incremental) coordinates

This cycle performs an inclined 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

The algebraic sign for the DEPTH cycle parameter 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 preset 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.

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

Define the approach in **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The tools approaches the contour starting point on a straight line

Cycle parameters

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

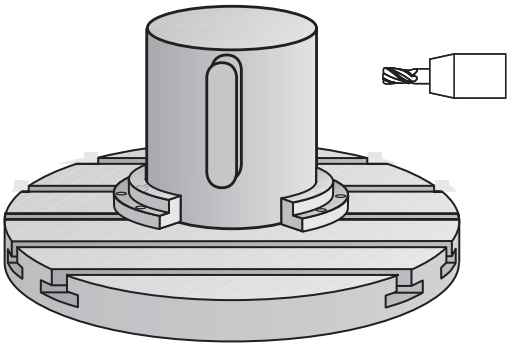
Example

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

11.4 CYLINDRICAL SURFACE ridge milling (Cycle 29, DIN/ISO: G129, option 1)

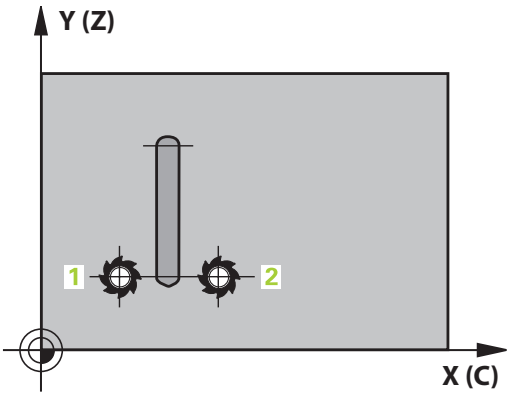
Cycle run

Refer to your machine manual!
The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



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

At the ends of the ridge, the control will always add a semi-circle whose radius corresponds to half the ridge width.



1 The control positions the tool above the starting point of machining. The control 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 to the left (**1**, RL = climb milling) or to the right of the ridge (**2**, RR = up-cut milling).
2 After the control has positioned the tool to the first plunging depth, the tool moves on a circular arc at the milling feed rate **Q12** tangentially to the ridge wall. A finishing allowance programmed for the side is taken into account.
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.

Please note while programming:

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** parameter (no. 201002), to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

This cycle performs an inclined 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

The algebraic sign for the DEPTH cycle parameter 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 preset 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 control will generate an error message. Switching of the kinematics may be required.

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

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

Cycle parameters

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

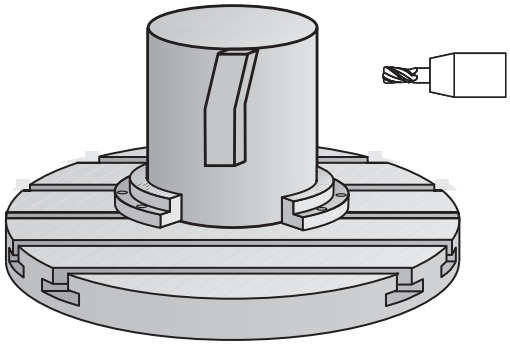
Example

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

11.5 CYLINDRICAL SURFACE CONTOUR (Cycle 39, DIN/ISO: G139, option 1)

Cycle run

Refer to your machine manual!
The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the developed surface of the cylinder. With this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the milled contour are always parallel to the cylinder axis.

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

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

1 The control positions the tool above the starting point of machining. The control locates the starting point next to the first point defined in the contour subprogram offset by the tool diameter
2 The control 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 **ConfigDatum**, **CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004) parameter)
3 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate **Q12** until the contour train is complete.
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.

Please note while programming:

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** parameter (no. 201002), to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

This cycle performs an inclined 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

The algebraic sign for the DEPTH cycle parameter 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 preset 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.

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

Define the approach in **apprDepCylWall** (no. 201004)

- CircleTangential: Tangential approach and departure
- LineNormal: The tools approaches the contour starting point on a straight line

Cycle parameters

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

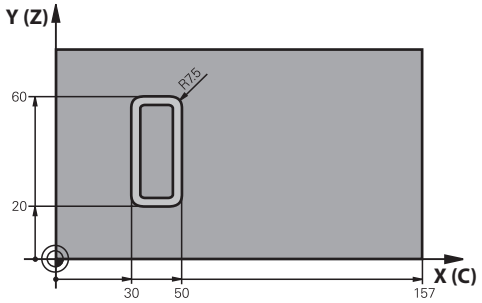
Example

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

11.6 Programming Examples

Example: Cylinder surface with Cycle 27

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

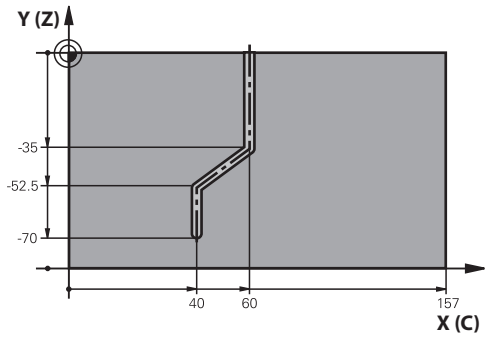


0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Positioning
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7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

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

Example: Cylinder surface with Cycle 28

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



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

12

Fixed Cycles: Contour Pocket with Contour Formula

12.1 SL cycles with complex contour formula

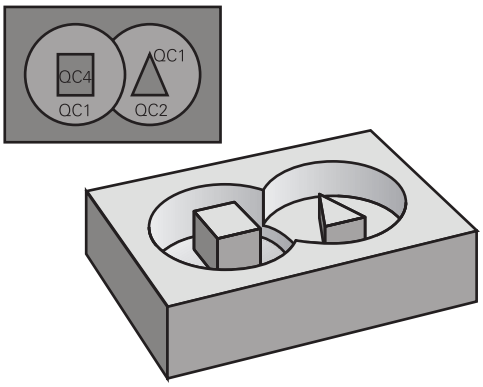
Fundamentals

Using SL cycles and the complex contour formula, you can combine subcontours (pockets or islands) to form complex contours. You define the individual subcontours (geometry data) in separate NC programs. In this way, any subcontour can be reused any number of times. The control calculates the complete contour from the selected subcontours, which you link 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** contour elements.

To use SL cycles with contour formulas, it is mandatory that your program is structured carefully. These cycles enable you to save frequently used contours in individual NC programs. Using the contour formula, you can connect the subcontours to define a complete contour and specify 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 control user interface. This function will serve as a basis for further development.



Program structure: Machining with SL cycles and complex contour formula

```
0 BEGIN PGM  KONTUR MM
...
5 SEL CONTOUR  "MODEL"
6 CYCL DEF 20  KONTUR-DATEN ...
8 CYCL DEF 22  RAEUMEN ...
9 CYCL CALL
...
12 CYCL DEF 23  SCHLICHTEN TIEFE ...
13 CYCL CALL
...
16 CYCL DEF 24  SCHLICHTEN SEITE ...
17 CYCL CALL
63 L  Z+250 R0  FMAX M2
64 END PGM  KONTUR MM
```

Properties of the subcontours

- The control assumes that each contour is a pocket. Thus, do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted—if they are programmed within the subcontours, they are also effective in the NC programs called subsequently. However, they need not be reset after the cycle call.
- Although the called NC programs can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the NC program.
- Subcontours can be defined with different depths according to your requirements.

Characteristics of the fixed cycles

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions—the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is 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 = “KREIS1“
2 DECLARE CONTOUR  QC2 = “KREISXY“
    DEPTH15
3 DECLARE CONTOUR  QC3 = “DREIECK“
    DEPTH10
4 DECLARE CONTOUR  QC4 =
    “QUADRAT“  DEPTH5
5 QC10 =  ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM  MODEL MM
```

```
0 BEGIN PGM  KREIS1 MM
1 CC  X+75 Y+50
2 LP  PR+45 PA+0
3 CP  IPA+360 DR+
4 END PGM  KREIS1 MM
```

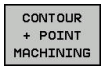
```
0 BEGIN PGM  KREIS31XY MM
...
...
```

Selecting an NC program with contour definitions

With the **SEL CONTOUR** function, you select an NC program with contour NC, from which the control extracts the contour descriptions:

Proceed as follows:

- Press the **SPEC FCT** key



- Press the **CONTOUR AND POINT MACHINING** soft key

- Press the **SEL CONTOUR** soft key.
- Enter the full name of the NC program with the contour definitions

- Alternatively, press the **SELECT FILE** soft key and select the program
- Confirm your input with the **END** key

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

Defining contour descriptions

Using the **DECLARE CONTOUR** function in your NC program, you enter the path for NC programs from which the control extracts the contour descriptions. In addition, you can select a separate depth for this contour description (FCL 2 function).

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

- Press the **CONTOUR AND POINT MACHINING** soft key

DECLARE CONTOUR

- Press the **DECLARE CONTOUR** soft key.
- Enter the number for the contour designator **QC**
- Press the **ENT** key
- Enter the full name of the NC program with the contour descriptions and confirm with the **ENT** key.

SELECT FILE

- Alternatively, press the **SELECT FILE** soft key and select the NC program
- Define a separate depth for the selected contour
- Press the **END** key

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).

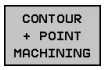
The control will only take different depths (**DEPTH**) into account if the elements overlap. In case of pure islands inside a pocket, this is not the case. Use a simple contour formula for this purpose.
Further information: "SL cycles with simple contour formula", Page 351

Entering a complex contour formula

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

Proceed as follows:

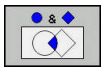
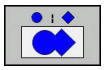
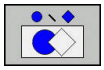
- Press the **SPEC FCT** key



- Press the **CONTOUR AND POINT MACHINING** soft key

- Press the **CONTOUR FORMULA** soft key
- Enter the number for the contour designator **QC**
- Press the **ENT** key

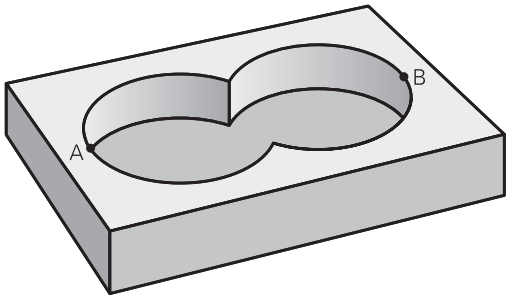
The control displays the following soft keys:

Soft key	Mathematical function
	**Intersected with** e.g. **QC10= QC1& QC5**
	**Joined with** e.g. **QC25= QC7| QC18**
	joined with, but without intersection e.g. **QC12 = QC5 ^ QC25**
	**without** e.g. **QC25 = QC1 \ QC2**
	Open parenthesis e.g. **QC12 = QC1 * (QC2 + QC3)**
	Close parenthesis e.g. **QC12 = QC1 * (QC2 + QC3)**
	Define single contour z. B. **QC12 = QC1**

Superimposed contours

By default, the control 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 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 control calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Contour description program 1: pocket A

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

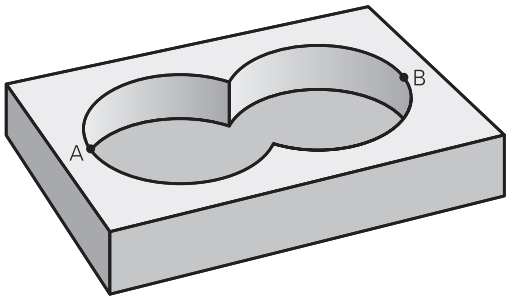
Contour description program 2: pocket B

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

Area of inclusion

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

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.



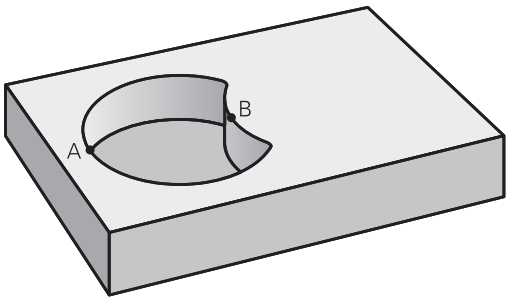
Contour definition program:

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

Area of exclusion

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

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the area B is subtracted from the area A with the **without** function.



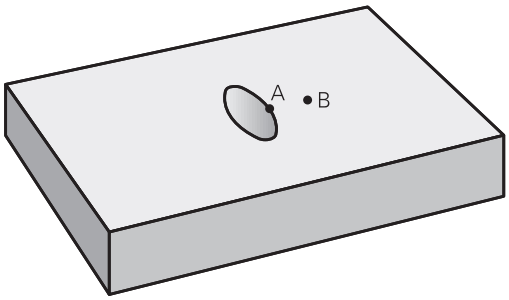
Contour definition program:

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

Area of intersection

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

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "intersection with" function.



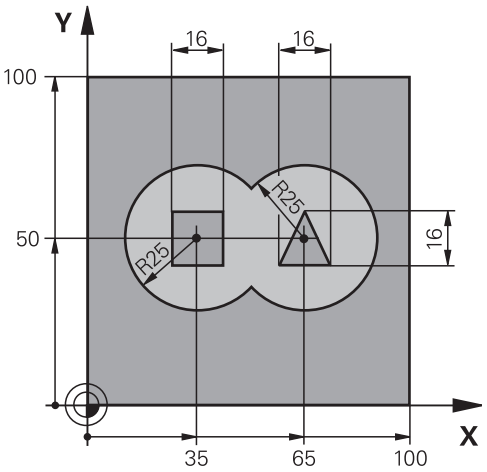
Contour definition program:

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

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", Page 260).

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	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: roughing cutter
4 L Z+250 R0 FMAX	Retract the tool
5 SEL CONTOUR "MODEL"	Define contour definition program
6 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	

7 CYCL DEF 22 ROUGH-OUT	Cycle definition: rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=+99999 ;RETRACTION FEED RATE	
Q401=100 ;FEED RATE FACTOR	
Q404=0 ;FINE ROUGH STRATEGY	
8 CYCL CALL M3	Cycle call: rough-out
9 TOOL CALL 2 Z S5000	Tool call: finishing cutter
10 CYCL DEF 23 FLOOR FINISHING	Cycle definition: floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE F. ROUGHNG	
Q208=+99999 ;RETRACTION FEED RATE	
11 CYCL CALL M3	Cycle call: floor finishing
12 CYCL DEF 24 SIDE FINISHING	Cycle definition: side finishing
Q9=+1 ;ROTATIONAL DIRECTION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE F. ROUGHNG	
Q14=+0 ;ALLOWANCE FOR SIDE	
13 CYCL CALL M3	Cycle call: side finishing
14 L Z+250 R0 FMAX M2	Retract the tool, end program
15 END PGM KONTUR MM	

Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	Contour definition program
1 DECLARE CONTOUR QC1 = “CIRCLE1”	Definition of the contour label for the "CIRCLE1" NC program
2 FN 0: Q1 =+35	Assignment of values for parameters used in PGM "CIRCLE31XY"
3 FN 0: Q2 =+50	
4 FN 0: Q3 =+25	
5 DECLARE CONTOUR QC2 = “CIRCLE31XY”	Definition of the contour label for the "CIRCLE3XY" NC program
6 DECLARE CONTOUR QC3 = “TRIANGLE”	Definition of the contour label for the "TRIANGLE" NC program
7 DECLARE CONTOUR QC4 = “SQUARE”	Definition of the contour label for the "SQUARE" NC program
8 QC10 = (QC 1 | QC 2) \ QC 3 \ QC 4	Contour formula
9 END PGM MODEL MM	

Contour description programs:

0 BEGIN PGM CIRCLE1 MM	Contour description program: circle at right
1 CC X+65 Y+50	
2 L PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE1 MM	

0 BEGIN PGM CIRCLE31XY MM	Contour description program: circle at left
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE31XY MM	

0 BEGIN PGM TRIANGLE MM	Contour description program: triangle at right
1 L X+73 Y+42 R0	
2 L X+65 Y+58	
3 L X+58 Y+42	
4 L X+73	
5 END PGM TRIANGLE MM	

0 BEGIN PGM SQUARE MM	Contour description program: square at left
1 L X+27 Y+58 R0	
2 L X+43	
3 L Y+42	
4 L X+27	
5 L Y+58	
6 END PGM SQUARE MM	

12.2 SL cycles with simple contour formula

Fundamentals

Using SL cycles and the simple contour formula, you can easily combine up to nine subcontours (pockets or islands) to form a particular contour. The control calculates the complete 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** contour 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  ROUGHING ...
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 control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted—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 control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions—the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling.

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

Entering a simple contour formula

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

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

- Press the **CONTOUR AND POINT MACHINING** soft key

CONTOUR DEF

- Press the **CONTOUR DEF** soft key
- Press the **ENT** key

> The control opens the dialog for entering the contour formula.

- Enter the first subcontour and confirm with the **ENT** key

POCKET

- Press the **POCKET** soft key

ISLAND

- Alternative: Press the **ISLAND** soft key
- Enter the second subcontour and confirm with the **ENT** key
- If needed, enter the depth of the second subcontour. Press the **ENT** key

> Carry on with the dialog as described above until you have entered all subcontours.

You can enter contours in the following ways:

Soft key	Function
CONTOUR <FILE>	Define the name of the contour
SELECT FILE	Alternatively, press the **SELECT FILE** soft key
CONTOUR <FILE>=QS	Define the number of a string parameter
CONTOUR LBL NR	Define the number of a label
CONTOUR LBL NAME	Define the name of the label
CONTOUR LBL QS	Define the number of a string parameter for a label

Always start the list of subcontours with the deepest pocket!

If the contour is defined as an island, the control 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 260).

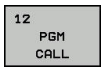
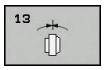
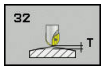
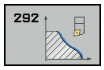
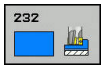
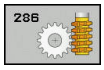
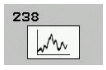
13

Cycles: Special Functions

13.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	9 DWELL TIME	357
	12 Program call	358
	13 Oriented spindle stop	359
	32 TOLERANCE	360
	291 COUPLING TURNING INTER-POLATION	364
	292 CONTOUR TURNING INTER-POLATION	371
	225 ENGRAVING of texts	382
	232 FACE MILLING	388
	285 DEFINE GEAR	395
	286 GEAR HOBBING	398
	287 GEAR SKIVING	404
	238 MEASURE MACHINE STATUS	409
	239 ASCERTAIN THE LOAD	411
	18 Thread cutting	413

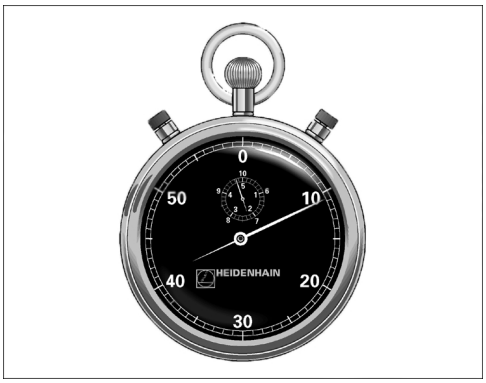
13.2 DWELL TIME (Cycle 9, ISO: G04)

Function

Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle becomes effective as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.



Example

```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

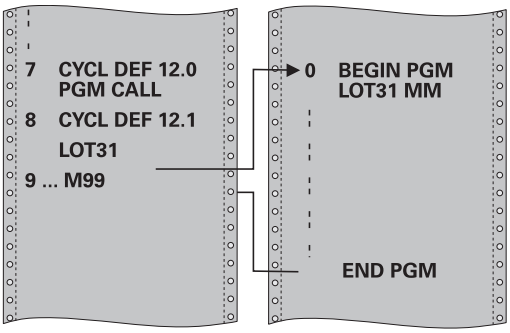
Cycle parameters

- **Dwell time in seconds**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

13.3 PROGRAM CALL (Cycle 12, ISO: G39)

Cycle function

NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.



Please note while programming:

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

The NC program you are calling must be stored in the internal memory of your control.

If the NC program you are defining to be a cycle is located in the same directory as the NC program you are calling it from, you need only enter the program name.

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

If you want to define an ISO program to be a cycle, add the .I file type to 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 NC program can also influence the calling NC program.

Cycle parameters

12 PGM CALL

- **Program name**: Enter the name of the NC program and, if necessary, the path where it is located, or
- Activate the file select dialog with the **SELECT** soft key. Select the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare program 50.h as a cycle and call it with M99

```
55 CYCL DEF 12.0  PGM CALL
56 CYCL DE 12.1  PGM TNC:
    \KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99
```

13.4 SPINDLE ORIENTATION (Cycle 13, ISO: G36)

Cycle function

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

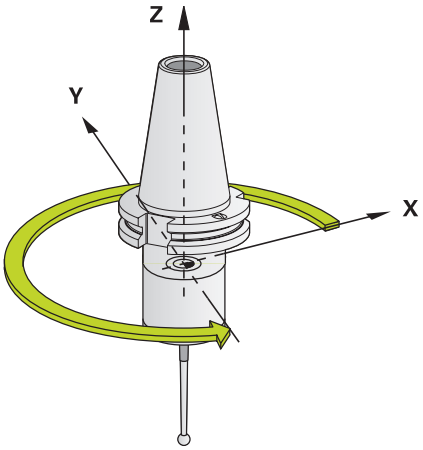
The control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

With M19 or M20, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program M19 or M20 without having defined Cycle 13 beforehand, the control positions the main spindle at an angle that has been set by the machine tool builder.



Example

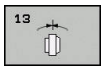
```
93 CYCL DEF 13.0  ORIENTATION
94 CYCL DEF 13.1  ANGLE 180
```

Please note while programming:

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

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

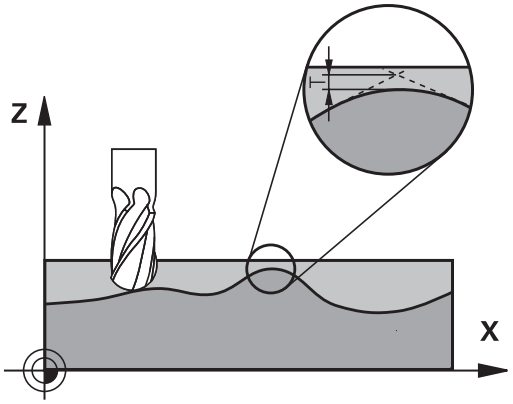


- **Angle of orientation**: Enter the angle with reference to the angle reference axis of the working plane. Input range: 0.0000° to 360.0000°

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

Cycle function

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



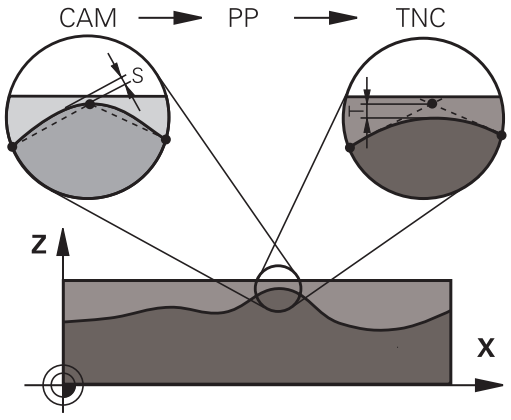
With the entries in Cycle 32 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the control has been adapted to the machine's characteristics.

The control automatically smooths the contour between any two contour elements (whether compensated or not). This means that 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 control automatically reduces the programmed feed rate so that the program can be executed at the fastest possible speed without jerking. **Even if the control does not move the axes with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the control 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 chord error defines the maximum point spacing of NC programs generated in a post processor (PP). If the chord error is less than or equal to the tolerance value **T** defined in Cycle 32, then the control 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!

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

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 control, but by the fact that, in order to machine the contour transitions very exactly, the control 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 NC program.

In a program with millimeters set as unit of measure, the control interprets the entered tolerance value **T** in millimeters. In an inch program it interprets it as inches.

If you load an NC program with Cycle 32 that contains only the **Tolerance value** T cycle parameter, the control inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine tool builder).

If Cycle 32 is active, the control shows the parameters defined for Cycle 32 on the **CYC** tab of the additional status display.

Reset

The control resets Cycle 32 if you do one of the following:

- Redefine Cycle 32 and confirm the dialog prompt for the **tolerance value** with **NO ENT**.
- Select a new NC program with the **PGM MGT** key.

After you have reset Cycle 32, the control reactivates the tolerance that was predefined by the machine parameters.

Keep the following in mind for 5-axis simultaneous machining!

NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more consistent. Additionally, in this cycle you can set a higher rotational axis tolerance **TA** (e.g. between 1° and 3°) for an even more constant feed-rate curve at the tool reference point (TCP).

For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotary axis tolerance. This contour damage depends on the possible tool tilting, tool radius and engagement depth of the tool.
With 5-axis hobbing with an end mill, you can calculate the maximum possible contour damage T directly from the cutter engagement length L and permissible contour tolerance TA:
T ~ K x L x TA K = 0.0175 [1/°]
Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

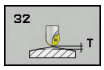
Sample formula for a toroid cutter:

When machining with a toroid cutter, the angle tolerance is very important.

$$Tw = \frac{180}{\pi * R} T_{32}$$

T_w: Angle tolerance in degrees
π: Circular constant (pi)
R: Major radius of the torus in mm
T_{32}: Machining tolerance in mm

Cycle parameters



- **Tolerance value T**: Permissible contour deviation in mm (or inches with inch programming). Input range: 0.0000 to 10.0000
  >0: If you enter a value greater than zero, the control will use the maximum permissible deviation you specify
  0: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine tool builder.
- **HSC MODE, Finishing=0, Roughing=1**: Activate filter:
  - Input value 0: **Milling with increased contour accuracy**. The control uses internally defined finishing filter settings.
  - Input value 1: **Milling at an increased feed rate**. The control 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 control 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 NC programs for more than one axis by entering a large tolerance value (e.g. 10°), because the control does not always have to position the rotary axis at the given nominal position. The tool orientation (position of the rotary axis with respect to the workpiece surface) will be adjusted. The position at the **T**ool **C**enter **P**oint (TCP) will be corrected automatically. For example, with a spherical cutter measured in its center and programmed based on the center path, there will be no adverse effects on the contour. Input range: 0.0000 to 10.0000
  >0: If you enter a value greater than zero, the control will use the maximum permissible deviation you specify.
  0: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine tool builder.

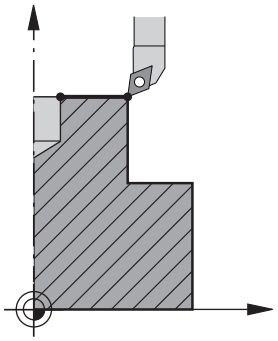
Example

```
95 CYCL DEF 32.0  TOLERANCE
96 CYCL DEF 32.1  T0.05
97 CYCL DEF 32.2  HSC-MODE:1 TA5
```

13.6 COUPLING TURNING INTERPOLATION (Cycle 291, DIN/ISO: G291, option 96)

Cycle run

Refer to your machine manual!
Option 96 must have been enabled.
This function must be enabled and adapted by the machine tool builder.



Cycle **291 COUPLG.TURNG.INTERP.** couples the tool spindle to the position of the linear axes, or cancels this spindle coupling. With interpolation turning, the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates **Q216** and **Q217**.

Cycle run if Q560=1:

1 The control first performs a spindle stop (M5).
2 The control 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.
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 or end of program/internal stop.)

Cycle run if Q560=0:

1 The control 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, parameters **Q336**, **Q216**, **Q217** are not relevant

Please note while programming!

After defining Cycle 291 and **CYCL CALL** program the operation you wish to perform. To describe the circular motions of the linear axes, you can use linear or polar coordinates, for example. An example is provided at the end of this section.

Further information: "Example: Interpolation turning with Cycle 291", Page 416

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

Your control 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.

The machine tool builder defines an M function for spindle orientation in the **CfgGeoCyclemStrobeOrient** machine parameter (no. 201005).
If the value is >0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine tool builder). The control waits until the oriented spindle stop has been completed.
If you enter –1, the control will perform the oriented spindle stop.
If you enter 0, no action will be taken.
The control will under no circumstances output M5.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle 291 is CALL-active.

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

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.

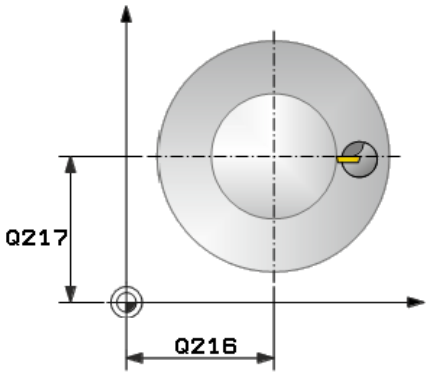
If Cycle **8 MIRRORING** is active, the control does **not** perform the cycle for interpolation turning.

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

Please keep in mind that the axis angle must be equal to the tilt angle before calling the cycle. Only then is correct coupling of the axes possible.

Cycle parameters

- **Q560 Spindle coupling (0=off, 1=on)?**: Specify whether the tool spindle is coupled to the position of the linear axes or not. If spindle coupling is active, the tool's cutting edge is oriented to the center of rotation.
  0: Spindle coupling off
  1: Spindle coupling on
- **Q336 Angle for spindle orientation?**: The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input range: 0.000 to 360.000
  Further information: "Defining the tool", Page 367
- **Q216 Center in 1st axis?** (absolute): Center of rotation in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Center of rotation in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q561 Convert from turning tool** (0/1): Only relevant if you define the turning tool in the turning tool table (toolturn.trn). With this parameter, you decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool.
  0: No change; the turning tool is interpreted as described in the turning tool table (toolturn.trn). In this case you may not use radius compensation **RR** or **RL**. Furthermore, you must describe the movement of the tool center path **TCP** without spindle coupling when programming. This kind of programming is much more difficult.
  1: The value XL from the turning tool table (toolturn.trn) will be interpreted as a radius R from a milling tool table. This makes it possible to use radius compensation **RR** and **RL** when programming your contour. This kind of programming is recommended.



Example

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

Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either activate (**Q560**=1) or deactivate (**Q560**=0) the COUPLG.TURNG.INTERP. cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

Q560=0: Disable the **COUPLG.TURNG.INTERP.** cycle!

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter **Q560**=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 your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece with inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  - Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

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

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

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

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

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

If you programmed parameter **Q561**=1, you must program the following in order to conclude the interpolation turning machining operation:

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

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

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

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + **Q336**
Interpolation turning, inside	7	ORI + **Q336** + 180
Interpolation turning, outside	7	ORI + **Q336** + 180
Interpolation turning, inside	1	ORI + **Q336**
Interpolation turning, outside	8	ORI + **Q336**
Interpolation turning, inside	8	ORI + **Q336**

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

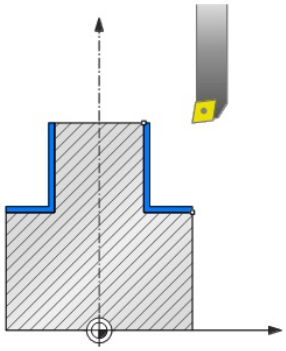
13.7 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, DIN/ ISO : G292, option 96)

Cycle run

Refer to your machine manual!

Option 96 must have been enabled.

This function must be enabled and adapted by the machine tool builder.

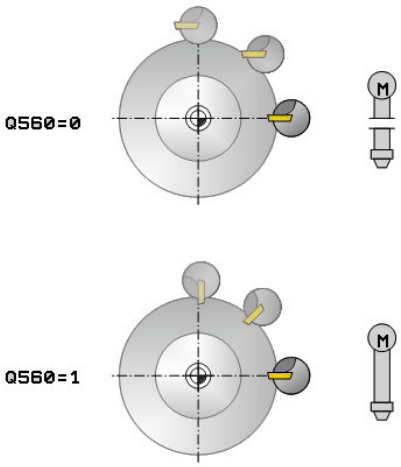


Cycle 292 **INTERPOLATION TURNING, CONTOUR FINISHING** couples the tool spindle to the positions of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. After executing this cycle, the control deactivates the spindle coupling again.

Before using Cycle 292, you first need to define the desired contour in a subprogram and refer to this contour with Cycle 14 or **SEL CONTOUR**. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. If you enter **Q560**=1, you can turn the contour and the cutting edge is oriented toward the circle center. If you enter **Q560**=0, you can mill the contour and the spindle is not oriented toward the circle center.

Cycle run, Q560=1: Contour turning

1 The control orients the tool spindle to the specified center of rotation. The specified angle **Q336** is taken into account. If an "ORI" value is given in the turning tool table (toolturn.trn), it is also taken into account.
2 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
3 The control positions the tool at the contour start radius **Q491**, taking the selected machining operation (inside/outside, **Q529**) and the set-up clearance to the side, **Q357**, into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4 The control uses the interpolation turning cycle to machine the defined contour. In interpolation turning, the linear axes of the working plane move on a circle, whereas the spindle axis follows, it is oriented perpendicularly to the surface.
5 At the end point of the contour, the control retracts the tool perpendicularly to the set-up clearance.
6 Finally, the control retracts the tool to the clearance height.
7 The control 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 will be performed. **Q336** is not taken into account
3 The control positions the tool at the contour start radius **Q491**, taking the selected machining operation (inside/outside, Q529) and the set-up clearance to the side, **Q357**, into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4 The control machines the defined contour using a rotating spindle (M3/M4). The principal axes of the working plane move on a circle, whereas the spindle axis does not follow.
5 At the end point of the contour, the control retracts the tool perpendicularly to the set-up clearance.
6 Finally, the control retracts the tool to the clearance height.

Please note while programming:

An example is provided at the end of this section, see Page 419.

NOTICE

Danger of collision!

There is a risk of collision between tool and workpiece. The control does not automatically extend the described contour by a set-up clearance! At the beginning of the machining operation, the control positions the tool at rapid traverse FMAX to the contour starting point!

- Program an extension of the contour in the subprogram
- 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.

With **Q560**=1, the control does not check whether the cycle is run with a rotating or stationary spindle. (Independent of **CfgGeoCycle** - **displaySpindleError** (no. 201002))

Your control 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.

The machine tool builder defines an M function for spindle orientation in the **CfgGeoCyclemStrobeOrient** machine parameter (no. 201005).
If the value is >0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine tool builder). The control waits until the oriented spindle stop has been completed.
If you enter –1, the control will perform the oriented spindle stop.
If you enter 0, no action will be taken.
The control will under no circumstances output M5.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The cycle is CALL-active.

Please note that it is not possible to define programmed finishing allowances via the **FUNCTION TURNDATA CORR-TCS(WPL)** function. Program a finishing allowance for your contour directly in the cycle or by specifying a tool compensation (DXL, DZL, DRS) in the tool table.

When programming, remember to use only positive radius values.

Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

Roughing operations with multiple passes are not possible in this cycle.

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour **Q491** plus the set-up clearance to the side **Q357**. If the control determines that the tool is too large, the NC program will be canceled.

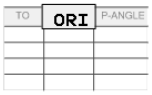
Please keep in mind that the axis angle must be equal to the tilt angle before calling the cycle. Only then is correct coupling of the axes possible.

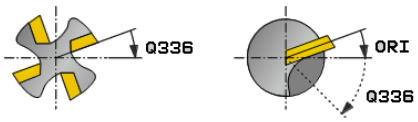
If Cycle **8 MIRRORING** is active, the control does **not** perform the cycle for interpolation turning.

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

Cycle parameters

- **Q560 Spindle coupling (0=off, 1=on)?**: Specify whether the spindle should be coupled or not.
  0: Spindle coupling off (mill the contour)
  1: Spindle coupling on (turn the contour)
- **Q336 Angle for spindle orientation?**: The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input range: 0.000 to 360.000
- **Q546 Reverse tool rotation direction?**: Direction of spindle rotation of the active tool:
  3: Clockwise rotation (M3)
  4: Counterclockwise rotation (M4)
- **Q529 Machining operation (0/1)?**: Specify whether an inside or outside contour will be machined:
  +1: Inside machining
  0: Outside machining
- **Q221 Oversize for surface?**: Allowance in the working plane. Input range: 0 to 99.9999
- **Q441 Infeed per revolution [mm/rev]?**: Dimension by which the control approaches the tool during one revolution. Input range: 0.001 to 99.999
- **Q449 Feed rate / cutting speed?** (mm/min): Feed rate relative to the contour starting point **Q491**. Input range: 0.1 to 99999.9. The feed rate of the tool's center path is adjusted according to the tool radius and **Q529 MACHINING OPERATION**. From these parameters, the control determines the programmed cutting speed at the diameter of the contour starting point.
  Q529=1: Feed rate of the tool's center path is reduced for inside machining
  Q529=0: Feed rate of the tool's center path is increased for outside machining





Example

63 CYCL DEF 292 CONTOUR.TURNG.INTRP.	
Q560=1	;SPINDLE COUPLING
Q336=0	;ANGLE OF SPINDLE
Q546=3	;CHANGE TOOL DIRECTN.
Q529=0	;MACHINING OPERATION
Q221=0	;SURFACE OVERSIZE
Q441=0.5	;INFEED
Q449=2000	;FEED RATE
Q491=0	;CONTOUR START RADIUS
Q357=2	;CLEARANCE TO SIDE
Q445=50	;CLEARANCE HEIGHT

- **Q491 Contour starting point (radius)?** (absolute value): Radius of the contour starting point (e.g. X coordinate, if tool axis is Z). Input range: 0.9999 to 99999.9999
- **Q357 Safety clearance to the side?** (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth Input range: 0 to 99999.9
- **Q445 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece; the tool retracts to this position at the end of the cycle. Input range: -99999.9999 to 99999.9999

Machining variants

Before using Cycle 292, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

Tool axis used	Axial coordinate	Radial coordinate
Z	Z	X
X	X	Y
Y	Y	Z

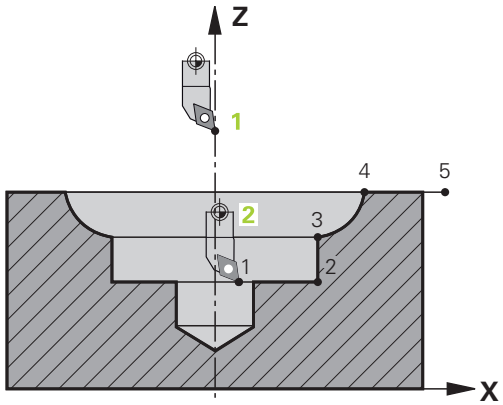
Example: If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius of the contour in X.

You can use this cycle for inside and outside machining. The following information illustrates some of the notes given in the "Please note while programming" section. You can also find an example in "Example: Interpolation Turning Cycle 292", Page 419

Inside machining

- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation!** Keep this in mind while describing the contour **2**
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control 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)**

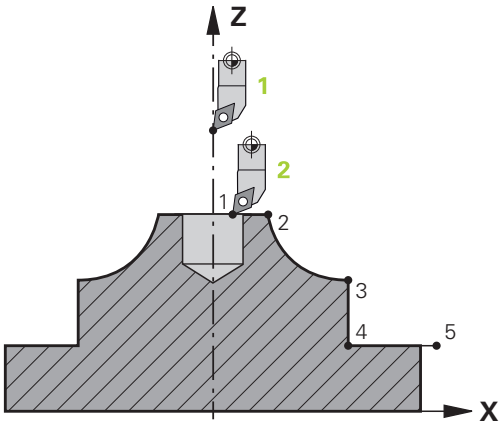
  When programming an inside contour, please also remember:
  - Program either monotonously increasing radial and axial coordinates, e.g. 1-5
  - Or program monotonously decreasing radial and axial coordinates, e.g. 5-1
  - Program inside contours with a radius greater than the tool radius.



Outside machining

- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! **2**
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control 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 monotonously increasing radial coordinates and monotonously decreasing axial coordinates, e.g. 1-5
  - Or program monotonously decreasing radial coordinates and monotonously increasing axial coordinates, e.g. 5-1
  - Program outside contours with a radius greater than 0.



Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either mill (**Q560**=0) or turn (**Q560**=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

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

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

NOTICE

Danger of collision!

Collision may occur between the tool holder and workpiece with inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

- ▶ Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

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

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.
- **Define a turning tool in the turning tool table (toolturn.trn)**

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

  The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + **Q336**
Interpolation turning, inside	7	ORI + **Q336** + 180
Interpolation turning, outside	7	ORI + **Q336** + 180
Interpolation turning, inside	1	ORI + **Q336**
Interpolation turning, outside	8,9	ORI + **Q336**
Interpolation turning, inside	8,9	ORI + **Q336**

You can use the following tool types for interpolation turning:

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

For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour **Q491** plus the set-up clearance to the side **Q357**. If the control determines that the tool is too large, the NC program will be canceled.

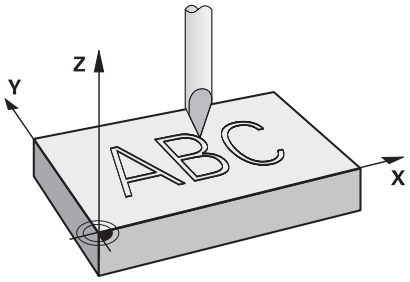
The following tool types cannot be used for interpolation turning: (error message "Function not possible with this tool type" is displayed)

- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

13.8 ENGRAVING (Cycle 225, 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 control 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 control 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 control retracts the tool to the 2nd set-up clearance.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

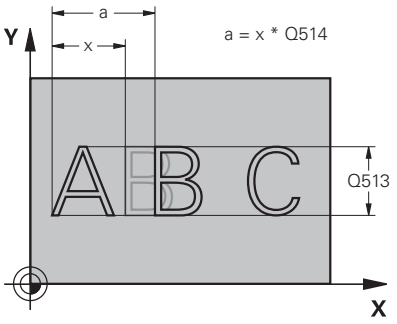
The text to be engraved can also be transferred with a string variable (**QS**).

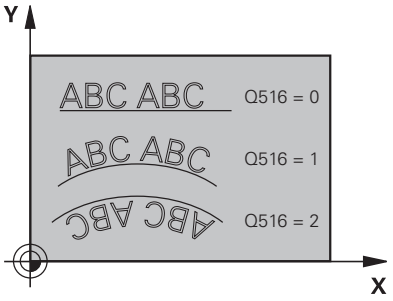
Parameter **Q347** influences the rotational position of the letters.
If **Q374**=0° to 180°, the characters are engraved from left to right.
If **Q374** is greater than 180°, the direction of engraving is reversed.

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

Cycle parameters

- **QS500 Engraving text?**: Text to be engraved, in quotation marks. Maximum input: 255 characters. Assignment of a string variable through the **Q** key of the numerical keypad. The **Q** key on the alphabetic keyboard represents normal text input. see "Engraving system variables", Page 386
- **Q513 Character height?** (absolute): Height of the characters to be engraved in mm. Input range: 0 to 99999.9999
- **Q514 Character spacing factor?**: The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the control if you program **Q514**=0. If **Q514** is not equal to 0, the control scales the space between the characters. Input range: 0 to 9.9999
- **Q515 Font?**: By default, the control uses the **DeJaVuSans** font.
- **Q516 Text on a line/on an arc(0/1)?**:
  Engrave the text in a straight line: Input = 0
  Engrave the text on an arc: Input = 1
  Engrave the text on an arc, circumferentially (not necessarily legible from below): Input = 2
- **Q374 Angle of rotation?**: Center angle if the text is to be engraved on an arc. Engraving angle when text is in a straight line. Input range: –360.0000 to +360.0000°
- **Q517 Radius of text on an arc?** (absolute): Radius of the arc in mm on which the control is to engrave the text Input range: 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q201 Depth?** (incremental): Distance between workpiece surface and engraving floor
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**





Example

```
62 CYCL DEF 225  ENGRAVING
    QS500="A"   ;ENGRAVING TEXT
    Q513=10     ;CHARACTER HEIGHT
    Q514=0      ;SPACE FACTOR
    Q515=0      ;FONT
    Q516=0      ;TEXT ARRANGEMENT
    Q374=0      ;ANGLE OF ROTATION
    Q517=0      ;CIRCLE RADIUS
    Q207=750    ;FEED RATE MILLING
    Q201=-0.5   ;DEPTH
    Q206=150    ;FEED RATE FOR PLNGNG
    Q200=2      ;SET-UP CLEARANCE
    Q203=+20    ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q367=+0     ;TEXT POSITION
    Q574=+0     ;TEXT LENGTH
```

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

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers:

! # $ % & ‘ ( ) * + , - . / : ; < = > ? @ [\ ] _ ß CE

The control uses the special characters **%** and **** for special functions. f you want to engrave these characters, enter them twice in the text to be engraved e. g. **%%**).

When engraving German umlauts, ß, ø, @, or the CE character, enter the character % before the character to be engraved:

Algebraic sign	Input
ä	%ae
ö	%oe
ü	%ue
Ä	%AE
Ö	%OE
Ü	%UE
ß	%ss
ø	%D
@	%at
CE	%CE

Characters that cannot be printed

Apart from text, you can also define certain non-printable characters for formatting purposes. Enter the special character **** before the non-printable characters.

The following formatting possibilities are available:

Character	Input
Line break	\n
Horizontal tab (the tab width is permanently set to 8 characters)	\t
Vertical tab (the tab width is permanently set to one line)	\v

Engraving system variables

In addition to the standard characters, you can engrave the contents of certain system variables. Precede the system variable with **%**.

You can also engrave the current date or time. Do do so, enter **%time<x>**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY. (Identical to the **SYSSTR ID321** function)

Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **%Time08**.

Character	Input
DD.MM.YYYY hh:mm:ss	%time00
D.MM.YYYY h:mm:ss	%time01
D.MM.YYYY h:mm	%time02
D.MM.YY h:mm	%time03
YYYY-MM-DD hh:mm:ss	%time04
YYYY-MM-DD hh:mm	%time05
YYYY-MM-DD h:mm	%time06
YY-MM-DD h:mm	%time07
DD.MM.YYYY	%time08
D.MM.YYYY	%time09
D.MM.YY	%time10
YYYY-MM-DD	%time11
YY-MM-DD	%time12
hh:mm:ss	%time13
h:mm:ss	%time14
h:mm	%time15

Engraving the name and path of an NC program

Use Cycle 225 to engrave the name and path of an NC program.

Define Cycle 225 as usual. Precede the engraved text with **%**.

It is possible to engrave the name or path of an active or called NC program. For this purpose, define **%main<x>** or **%prog<x>**. (Identical to the **ID10010 NR1/2** function)

The following formatting possibilities are available:

Character	Input	Engraved text
Full path of the active NC program	**%main0**	e.g. **TNC:\MILL.h**
Path to the directory of the active NC program	**%main1**	e.g. **TNC:**
Name of the active NC program	**%main2**	e.g. **MILL**
File type of the active NC program	**%main3**	e.g. **.H**
Full path of the called NC program	**%prog0**	e.g. **TNC:\HOUSE.h**
Path to the directory of the called NC program	**%prog1**	e.g. **TNC:**
Name of the called NC program	**%prog2**	e.g. **HOUSE**
File type of the active NC program	**%prog3**	e.g. **.H**

Engraving the counter reading

You can engrave the current counter reading found in the MOD menu with Cycle 225.

For this purpose program Cycle 225 as usual and enter e.g. the following for the text to be engraved: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is of nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following: 000000003

In the Test Run operating mode, the control only simulates the counter reading you specified directly in the NC program. The counter reading from the MOD menu is not taken into account.

In the SINGLE BLOCK and FULL SEQ. operating modes, the control will take the counter reading from the MOD menu into account.

13.9 FACE MILLING (Cycle 232, ISO: G232)

Cycle run

With Cycle 232, you can 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 control positions the tool at rapid traverse **FMAX** to the starting point **1** using positioning logic: If the current position in the spindle axis is further away from the workpiece 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 it 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 set-up 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 at the programmed feed rate for milling to the end point **2**. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.

4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.

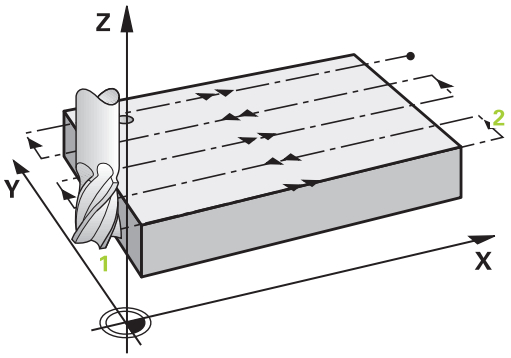
5 The tool then moves back in the direction of the starting point **1**.

6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.



Strategy Q389=1

3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **at the edge** of the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.

4 The control 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 pass 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 completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=2

3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.

4 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves it at the pre-positioning feed rate directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.

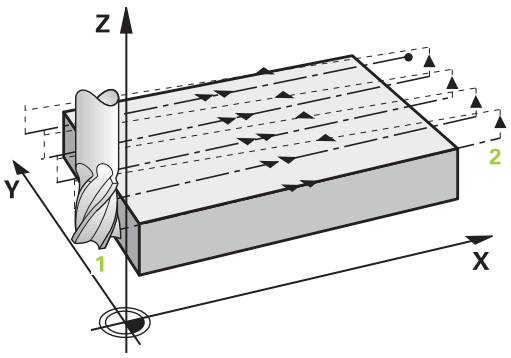
5 The tool then returns to the current infeed depth and moves in the direction of end point **2**

6 The process is repeated until the programmed surface has been machined completely. At the end of the last pass, the tool plunges to the next machining depth.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.



Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

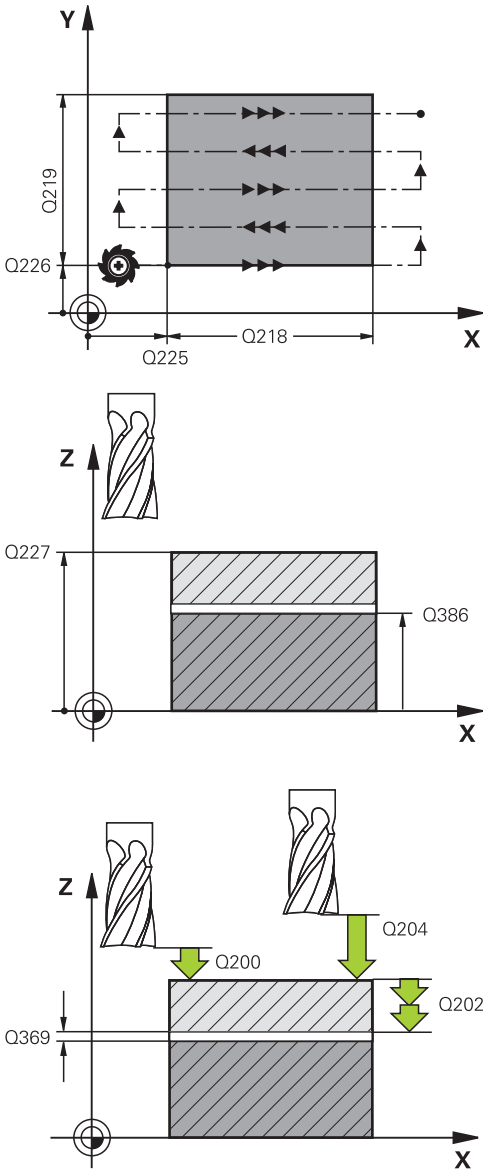
Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).

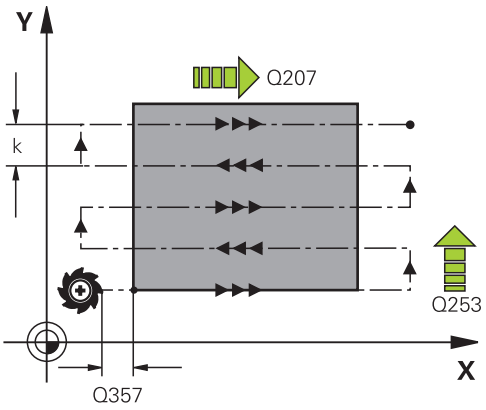
Program **Q227** greater than **Q386**. The control will otherwise display an error message.

Cycle parameters

- **Q389 Machining strategy (0/1/2)?**: Determine how the control should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
- **Q225 Starting point in 1st axis?** (absolute): Starting point coordinate of the surface to be machined in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q226 Starting point in 2nd axis?** (absolute): Starting point coordinate of the surface to be machined in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range: –99999.9999 to 99999.9999
- **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to which the surface is to be face-milled. Input range: –99999.9999 to 99999.9999
- **Q218 First side length?** (incremental): Length of the surface to be machined in the principal axis of the working plane. Use the algebraic sign to specify the direction of the first milling path referencing the **starting point in the 1st axis**. Input range: –99999.9999 to 99999.9999
- **Q219 Second side length?** (incremental): Length of the surface to be machined in the secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referencing the **STARTNG PNT 2ND AXIS**. Input range: –99999.9999 to 99999.9999
- **Q202 Maximum plunging depth?** (incremental): **Maximum** infeed per cut. The control calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range: 0 to 99999.9999



- **Q370 Max. path overlap factor?**: **Maximum** stepover factor k. The control 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. cutter radius when using a face-milling cutter), the control reduces the stepover accordingly. Input range: 0.1 to 1.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range: 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range: 0 to 99999.9999, alternatively **FAUTO**, **FU**, **FZ**
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (**Q389**=1), the control uses the cross feed rate for milling **Q207**. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy **Q389**=2, the control moves the tool to the set-up clearance above the current plunging depth to the starting point of the next pass. Input range: 0 to 99999.9999
- **Q357 Safety clearance to the side?** (incremental) Parameter **Q357** influences the following situations:
  Approaching the first plunging depth: Q357 is the lateral distance from the tool to the workpiece
  Roughing with the milling strategies Q389=0 to 3: The surface to be machined in the **Q350 MILLING DIRECTION** is increased by the value from **Q357** if there is no limit set in this direction
  Side finishing: The paths are extended by the value in **Q357** in the **Q350 MILLING DIRECTION**
  Input range: 0 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**



Example

```
71 CYCL DEF 232  FACE MILLING
   Q389=2      ;STRATEGY
   Q225=+10    ;STARTNG PNT 1ST AXIS
   Q226=+12    ;STARTNG PNT 2ND AXIS
   Q227=+2.5   ;STARTNG PNT 3RD AXIS
   Q386=-3     ;END POINT 3RD AXIS
   Q218=150    ;FIRST SIDE LENGTH
   Q219=75     ;2ND SIDE LENGTH
   Q202=2      ;MAX. PLUNGING DEPTH
   Q369=0.5    ;ALLOWANCE FOR FLOOR
   Q370=1      ;MAX. OVERLAP
   Q207=500    ;FEED RATE MILLING
   Q385=800    ;FINISHING FEED RATE
   Q253=2000   ;F PRE-POSITIONING
   Q200=2      ;SET-UP CLEARANCE
   Q357=2      ;CLEARANCE TO SIDE
   Q204=2      ;2ND SET-UP CLEARANCE
```

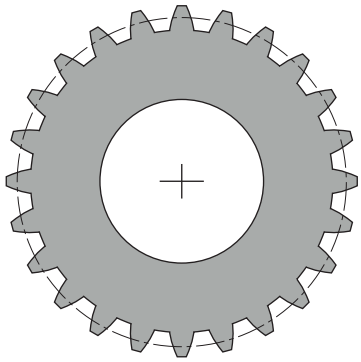
13.10 Gear manufacturing fundamentals (option 157)

Fundamentals

Refer to your machine manual!

Option 157 must have been enabled.

This function must be enabled and adapted by the machine tool builder.



For the cycles, option 157 Gear Cutting is required. If you would like to use these cycles in turning mode, you also need option 50. In milling mode, the tool spindle is the master spindle, in turning mode, it is the workpiece spindle. The other spindle is called slave spindle. Depending on the operating mode, you program the speed or the cutting speed with a **TOOL CALL S** or **FUNCTION TURNDATA SPIN**.

To orient the I-CS coordinate system, Cycles 286 and 287 use the precession angle that is also affected by Cycles 800 and 801 in turning mode. At the end of the cycle, the control resets the precession angle to its state at the beginning of the cycle. If one of these cycles is aborted, the precession angle will also be reset.

The axis crossing angle is the angle between workpiece and tool. It results from the angle of inclination of the tool and the angle of inclination of the gear. Based on the required axis crossing angle, Cycles 286 and 287 calculate the required inclination of the rotary axis at the machine. The cycles will always position the first rotary axis starting from the tool.

The gear itself will first be described in Cycle 285 **DEFINE GEAR**. Then, program Cycle 286 **GEAR HOBBING** or Cycle 287 **GEAR SKIVING**.

Program the following:

- Call a tool with **TOOL CALL**
- Select turning or milling mode with **FUNCTION MODE TURN / MILL**
- Spindle direction of rotation, e.g. **M3** or **M303**
- If required, program the **CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM** cycle
- Perform pre-positioning for the cycle depending on your selection of **MILL** or **TURN**
- Define the **CYCL DEF 285 DEFINE GEAR** cycle
- Define the **CYCL DEF 286 GEAR HOBBING** or **CYCL DEF 287 GEAR SKIVING** cycle.

Please note while programming!

NOTICE

Danger of collision!

If you do not pre-position the tool to a safe position, a collision between tool and workpiece (fixtures) may occur during tilting.

- Pre-position the tool to a safe position

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q200**!

- Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

Before calling the cycle, set the preset to the center of rotation of the workpiece spindle.

Please note that the slave spindle will continue to rotate after the end of the cycle. If you want to stop the spindle before the end of the program, make sure to program a corresponding M function.

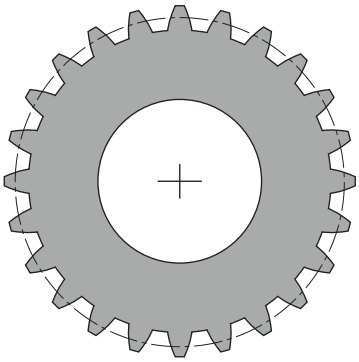
All machining feed rates are given in mm/rev. of the tool spindle.

The cycles automatically define the direction and path for a **LiftOff**. This function must be enabled by the machine tool builder. In addition, **LiftOff** must be permitted for the respective tool.

13.11 DEFINE GEAR (Cycle 285, DIN/ISO: G285, option 157)

Cycle run

With Cycle 285 **DEFINE GEAR**, you describe the geometry of the gearing system. To describe the tool, use Cycle 286 **GEAR HOBBING** or Cycle 287 for **GEAR SKIVING** and the tool table (TOOL.T).



Please note while programming!

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

You must specify values for module and number of teeth. If the outside diameter (diameter of the addendum circle) and the tooth height are defined as 0, normal running gears (DIN 3960) will be machined. If you want to machine gearing systems that differ from this standard, define the corresponding geometry by specifying the outside diameter **Q542** and the tooth height **Q563**.

Define the tool as a milling cutter in the tool table.

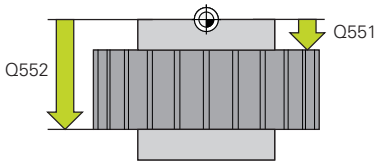
If the algebraic signs of the two input parameters **Q541** and **Q542** are contradictory, the cycle will be aborted with an error message.

This cycle is DEF-active. The values of these Q parameters will only be read when a CALL-active machining cycle is executed. If you overwrite these input parameters after the cycle definition and before calling the machining cycle, the gear geometry will be modified.

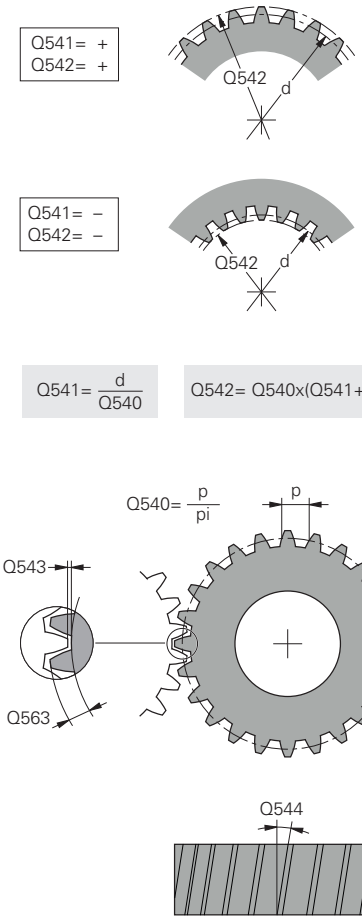
The two cycle parameters, **Q541 NUMBER OF TEETH** and **Q542 OUTSIDE DIAMETER** must have the same algebraic sign. Otherwise, an error message will be issued.

Cycle parameters

- **Q551 Starting point in Z?**: Starting point in Z for gear hobbing. Input range: -99999.9999 to 99999.9999
- **Q552 End point in Z?**: End point in Z for gear hobbing. Input range: -99999.9999 to 99999.9999
- **Q540 Module?**: Define the gear: Module of the gear wheel. Input range: 0 to 99.9999
- **Q541 Number of teeth?**: Number of teeth. This parameter depends on **Q542**.
  +: If the number of teeth is positive, and at the same time the parameter **Q542** is positive, then it is an external gear
  -: If the number of teeth is negative, and at the same time the parameter **Q542** is negative, then it is an internal gear
  Input range: –9999.9999 to +9999.9999
- **Q542 Outside diameter?**: Outside diameter of the gear wheel. This parameter depends on **Q541**.
  +: If the outside diameter is positive, and at the same time the parameter **Q541** is positive, then it is an external gear
  -: If the outside diameter is negative, and at the same time the parameter **Q541** is negative, then it is an internal gear
  Input range: –9999.9999 to +9999.9999



- **Q563 Tooth height?** Distance from the tooth trough to the tooth tip. Input range: 0 to 999.9999
- **Q543 Trough-to-tip clearance?**: Define the gear: Distance between the tip circle of the gear to be cut and the root circle of the mating gear. Input range: 0 to 9.9999
- **Q544 Angle of inclination?**: Define the gear: Angle by which teeth of a helical gear are inclined relative to the direction of the axis (For straight-cut gears this angle is 0°.) Input range: -60 to +60



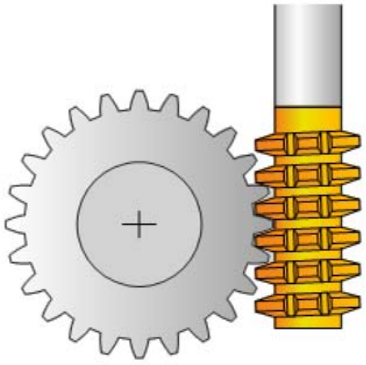
Example

```
63 CYCL DEF 285 DEFINE GEAR
    Q551=0          ;STARTING POINT IN Z
    Q552=-10        ;END POINT IN Z
    Q540=1          ;MODULE
    Q541=+10        ;NUMBER OF TEETH
    Q542=0          ;OUTSIDE DIAMETER
    Q563=0          ;TOOTH HEIGHT
    Q543=+0.17;TROUGH-TIP CLEARANCE
    Q544=0          ;ANGLE OF INCLINATION
```

13.12 GEAR HOBBING (Cycle 286, DIN/ISO: G286, option 157)

Application

With Cycle 286 **GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. 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 movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.



Cycle call

1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate **FMAX**. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the **FMAX** feed rate. 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 control then tilts the working plane at the feed rate **Q253**
4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane
5 The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6 The control now moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552**+**Q200** (**Q551** and **Q552** are defined in Cycle 285).
Further information: "DEFINE GEAR (Cycle 285, DIN/ISO: G285, option 157)", Page 395
7 When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally, the control retracts the tool to the clearance height **Q260** at the feed rate **FMAX**.

Please note while programming!

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

- Make sure to retract the tool before changing the position of the tilting axis

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

The cycle is CALL-active.

In order to ensure constant engagement of the cutting edge of a tool, you need to define a very small path in cycle parameter **Q554 SYNCHRONOUS SHIFT**.

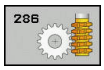
In turning mode, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM** before calling Cycle 286.

Avoid master spindle speeds of less than 6 rpm in turning mode. Otherwise, it is not possible to reliably use a feed rate in mm/rev. If you need to use a lower master spindle speed, rather use the cycle in milling mode than in turning mode.

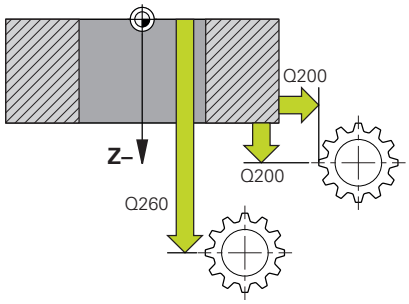
Make sure to program the direction of rotation of the master spindle before the cycle start.

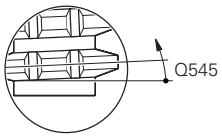
If you program **FUNCTION TURNDATA SPIN VCONST:OFF S15**, the spindle speed of the tool is calculated as **Q541** x S. With **Q541**=238 and S=15, this would result in a tool spindle speed of 3570 rpm.

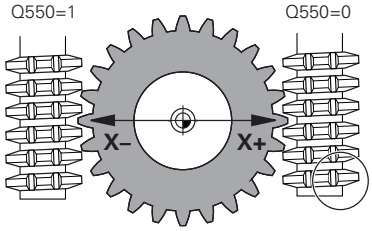
Cycle parameters

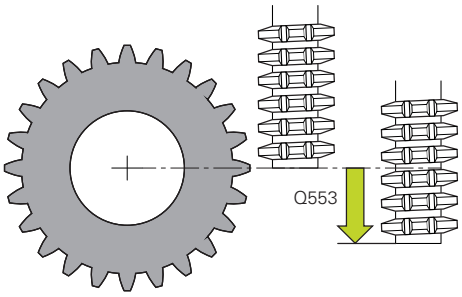


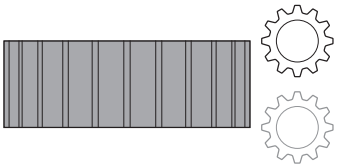
- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q200 Set-up clearance?** (incremental): Distance for retraction and pre-positioning. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: –99999.9999 to 99999.9999
- **Q545 Tool lead angle?**: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range: –60.0000 to +60.0000
- **Q546 Reverse spindle rotation dir.?**: Change direction of rotation of the slave spindle:
  0: The direction of rotation is not changed
  1: The direction of rotation is changed
  Further information: "Verifying and changing directions of rotation of the spindles", Page 403
- **Q547 Angle offset of tool spindle?**: Angle by which the control rotates the workpiece at the beginning of the cycle. Input range -180.0000 to +180.0000
- **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
  0: Positive machining side of the principal axis in the I-CS
  1: Negative machining side of the principal axis in the I-CS
- **Q533 Preferred dir. of incid. angle?**: Selection of alternative inclination options. The inclination angle you define is used by the control to calculate the appropriate position of the tilting axes on your machine. In general, there are always two possible solutions. Using parameter **Q533**, configure which of the solution options the control should apply:
  0: Option with the shortest distance from the current position
  –1: Option in the range between 0° and –179.9999°
  +1: Option in the range between 0° and +180°
  –2: Option in the range between –90° and –179.9999°
  +2: Option in the range between +90° and +180°

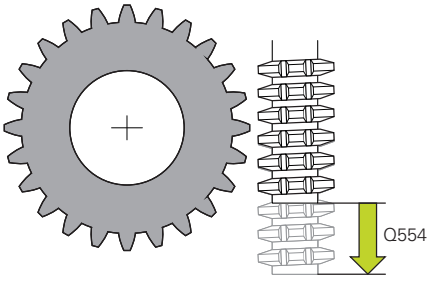












- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control carries out a compensation movement with the linear axes
  2: Automatically position the tilting axis without orienting the tool tip (TURN)
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Entry in mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q553 TOOL:L offset, machining start?** (incremental): Define the length offset (L OFFSET) from which the tool will machine. The tool will be offset in the longitudinal direction by this value. Input range: 0 to 999.9999
- **Q554 Path for synchronous shift?**: Define by which distance the gear hob will be offset in its axial direction during machining. This way, tool wear can be distributed over this area of the cutting edges. For helical gears, it is thus possible to limit the cutting edges used for machining. Entering 0 deactivates the synchronous shift function.
  Input range: -99.9999 to +99.9999
- **Q548 Tool shift for roughing?**: Specify the number of cutting edges by which the control will shift the roughing tool in its axial direction. The shift will be performed incrementally, referencing parameter **Q553**. Entering 0 deactivates the shift function. Input range: –99 to +99
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q488 Feed rate for plunging**: Feed rate for tool infeed. The control interprets the feed rate in mm per revolution. Input range: 0 to 99999.999 alternatively **FAUTO**, **PREDEF**

Example

```
63 CYCL DEF 286  GEAR HOBBING
   Q215=0       ;MACHINING OPERATION
   Q200=+2      ;SET-UP CLEARANCE
   Q260=+100    ;CLEARANCE HEIGHT
   Q545=0       ;TOOL LEAD ANGLE
   Q546=0       ;CHANGE ROTATION DIR.
   Q547=0       ;ANG. OFFSET, SPINDLE
   Q550=1       ;MACHINING SIDE
   Q533=0       ;PREFERRED DIRECTION
   Q530=2       ;INCLINED MACHINING
   Q253=750     ;F PRE-POSITIONING
   Q553=10      ;TOOL LENGTH OFFSET
   Q554=0       ;SYNCHRONOUS SHIFT
   Q548=0       ;ROUGHING SHIFT
   Q463=1       ;MAX. CUTTING DEPTH
   Q488=0.3     ;PLUNGING FEED RATE
   Q478=0.3     ;ROUGHING FEED RATE
   Q483=0.4     ;OVERSIZE FOR DIAMETER
   Q505=0.2     ;FINISHING FEED RATE
   Q549=0       ;FINISHING SHIFT
```

- **Q478 Roughing feed rate?**: Feed rate during roughing. The control interprets the feed rate in mm per revolution. Input range: 0 to 99999.999; alternatively **FAUTO**, **PREDEF**
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q505 Finishing feed rate?**: Feed rate during finishing. The control interprets the feed rate in mm per revolution. Input range: 0 to 99999.999; alternatively **FAUTO**, **PREDEF**
- **Q549 Tool shift for finishing?**: Specify the number of cutting edges by which the control will shift the finishing tool in its longitudinal direction. The shift will be performed incrementally, referencing parameter **Q553**. Entering 0 deactivates the shift function. Input range: –99 to +99

Verifying and changing directions of rotation of the spindles

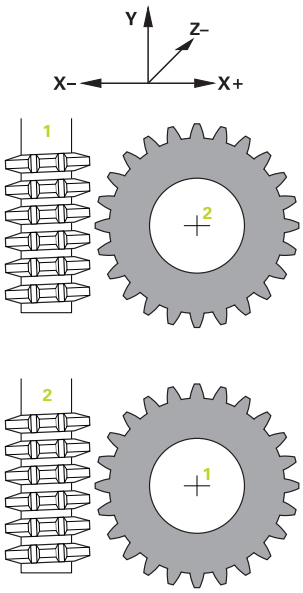
Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Changing the direction of rotation in milling mode:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Changing the direction of rotation in turning mode:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.



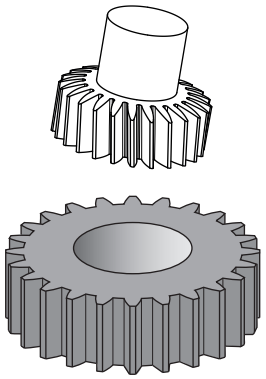
If required, define a low spindle speed to make sure that the direction of rotation is correct.

13.13 GEAR SKIVING (Cycle 287, DIN/ISO: G287, option 157)

Application

With Cycle 287 **GEAR SKIVING**, you can machine external cylindrical gears or helical gears with any angles. Cutting takes place on the one hand by the axial feeding of the tool and on the other hand through the rolling motion.

You can select the machining side in the cycle. The machining process for gear skiving is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.



Cycle call

1 The control positions the tool in the tool axis at the 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 control positions the tool in X at the feed rate **FMAX** to a safe coordinate. 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 control then tilts the working plane at the feed rate **Q253**

4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane

5 After that the control positions the tool in the tool axis at the feed rate **Q253** to the safety clearance **Q200**

6 The control then traverses the approach length. This distance is calculated by the control. The approach length is the distance from the initial scratch to the complete plunging depth.

7 The control rolls the tool over the workpiece to be geared in longitudinal direction at the defined feed rate. During the initial infeed of the cut **Q586** the control moves at the initial feed rate **Q588**. The control then uses intermediate values for the infeed and feed rate of the next cuts. The control calculates these values itself. However, the intermediate values for the feed rate depend on the factor for feed-rate adaptation **Q580**. When the control arrives at the last infeed **Q587**, it performs the last cut with the feed rate **Q589**

8 The area to be machined is limited by the starting point in Z **Q551**+**Q200** and the end point in Z **Q552** (**Q551** and **Q552** are defined in Cycle 285) The approach length must be added to the starting point. Its purpose is to prevent the tool from plunging into the workpiece all the way to the machining diameter. The control calculates this distance itself.

9 At end of machining the control traverses the idle travel distance. The idle travel distance serves to completely machine the gearing system to the end point. The control also calculates this distance itself

10 When the control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point

11 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX

Please note while programming!

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

- Make sure to retract the tool before changing the position of the tilting axis

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

The cycle is CALL-active.

In turning mode, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM** before calling Cycle 287.

Make sure to program the direction of rotation of the master spindle before the cycle start.

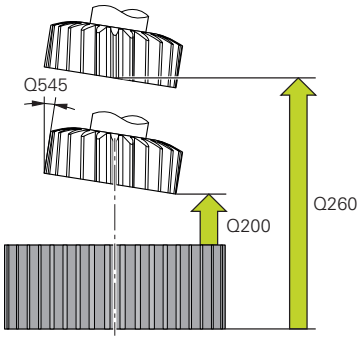
The larger the factor in **Q580 FEED-RATE ADAPTION**, the earlier the control will adapt the feed rate to the feed rate for the last cut. The recommended value is 0.2.

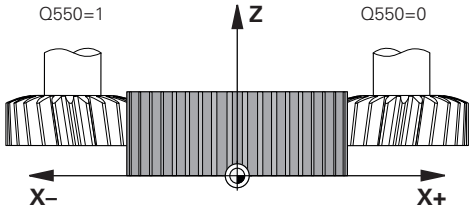
When defining the tool, make sure to specify the number of cutting edges as indicated in the tool table.

The speed ratio between tool and workpiece results from the number of teeth of the gear wheel and the number of cutting edges of the tool.

Cycle parameters

- **Q240 Number of cuts?** Number of cuts to the final depth
  0: The minimum number of cuts is determined automatically
  1: One cut
  2: Two cuts, here only the infeed for the first cut **Q586** is considered. The infeed for the last cut **Q587** is not considered here
  3 to 99999: Programmed number of cuts
- **Q584 Number of the first cut?**: Specify which cut number the control performs first. Input range: 1 to 999
- **Q585 Number of the last cut?**: Specify at which number the control should perform the last cut. Input range: 1 to 999
- **Q200 Set-up clearance?** (incremental): Distance for retraction and pre-positioning. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: –99999.9999 to 99999.9999
- **Q545 Tool lead angle?**: Describe the tool: Angle of the tooth sides of the gear skiver. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range: –60.0000 to +60.0000
- **Q546 Reverse spindle rotation dir.?**: Change direction of rotation of the slave spindle:
  0: The direction of rotation is not changed
  1: The direction of rotation is changed
  Further information: "Verifying and changing directions of rotation of the spindles", Page 408
- **Q547 Angle offset of tool spindle?**: Angle by which the control rotates the workpiece at the beginning of the cycle. Input range -180.0000 to +180.0000
- **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
  0: Positive machining side of the principal axis in the I-CS
  1: Negative machining side of the principal axis in the I-CS





Example

```
63 CYCL DEF 287 GEAR SKIVING
    Q240=0      ;NUMBER OF CUTS
    Q584=+1     ;NO. OF FIRST CUT
    Q585=+999   ;NO. OF LAST CUT
    Q200=2      ;SET-UP CLEARANCE
    Q260=+100   ;CLEARANCE HEIGHT
    Q545=0      ;TOOL LEAD ANGLE
    Q546=0      ;CHANGE ROTATION DIR.
    Q547=0      ;ANG. OFFSET, SPINDLE
    Q550=+1     ;MACHINING SIDE
    Q533=0      ;PREFERRED DIRECTION
    Q530=+2     ;INCLINED MACHINING
    Q253=+750   ;F PRE-POSITIONING
    Q586=+1     ;FIRST INFEED
    Q587=+0.1   ;LAST INFEED
    Q588=+0.2   ;FIRST FEED RATE
    Q589=+0.05  ;LAST FEED RATE
    Q580=+0.2   ;FEED-RATE ADAPTION
```

- **Q533 Preferred dir. of incid. angle?**: Selection of alternative inclination options. The inclination angle you define is used by the control to calculate the appropriate position of the tilting axes on your machine. In general, there are always two possible solutions. Using parameter **Q533**, configure which of the solution options the control should apply:
  0: Option with the shortest distance from the current position
  –1: Option in the range between 0° and –179.9999°
  +1: Option in the range between 0° and +180°
  –2: Option in the range between –90° and –179.9999°
  +2: Option in the range between +90° and +180°
- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control carries out a compensation movement with the linear axes
  2: Automatically position the tilting axis without orienting the tool tip (TURN)
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Entry in mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q586 Infeed for first cut?** (incremental): Infeed for the first cut. Input range: 0.001 to 99.999
- **Q587 Infeed for last cut?** (incremental): Infeed for the last cut. Input range: 0.001 to 99.999
- **Q588 Feed rate for first cut?**: Feed rate for the first cut. The control interprets the feed rate as millimeters per revolution. Input range: 0.001 to 99.999
- **Q589 Feed rate for last cut?**: Feed rate for the last cut. The control interprets the feed rate as millimeters per revolution. Input range: 0.001 to 99.999
- **Q580 Factor for feed-rate adaptation?**: Since it is necessary to reduce the feed rate with increasing cut numbers, you can use this factor to define a feed rate reduction. The greater the value, the earlier the control will adapt the feed rates to match the last feed rate. Input range: 0.000 to 1.000

Verifying and changing directions of rotation of the spindles

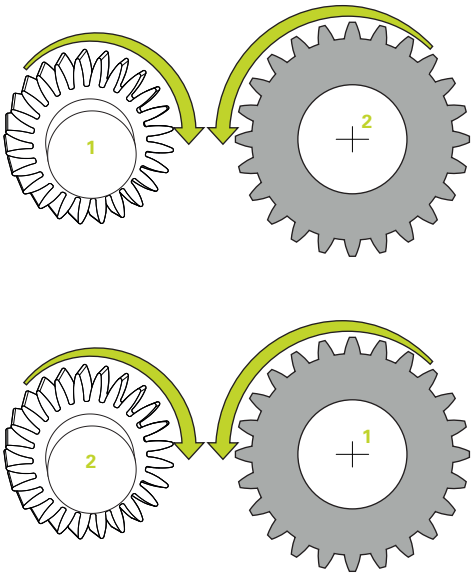
Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Changing the direction of rotation in milling mode:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Changing the direction of rotation in turning mode:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.



If required, define a low spindle speed to make sure that the direction of rotation is correct.

13.14 MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)

Application

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.
Option 155 (**Component Monitoring**) is required for Cycle 238.

During their lifecycle, the machine components which are subject to loads (e.g. guides, ball screws, ...) become worn and thus, the quality of the axis movements deteriorates. This, in turn, affects the production quality.

Using **Component Monitoring** (option 155) and Cycle 238, the control is able to measure the current machine status. Thus, deviations from the shipping condition of the machine that are due to wear and aging can be measured. The measurement results are stored in a text file that is readable for the machine tool builder. He can read the data, evaluate it and react appropriately by scheduling preventive maintenance. This way, unwanted machine downtimes can be prevented!

The machine tool builder can define warning and error thresholds for the measured values and optionally specify error reactions.

Cycle run

Parameter Q570=0

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are effective

Your machine tool builder defines in detail how the axes will move.

Parameter Q570=1

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are **not** effective
3 In the **MON Detail** status tab, you can select the monitoring tasks to be displayed
4 This diagram allows you to watch how close the components are to a warning or error threshold

Further information: User's Manual for Setup, Testing and Running NC programs

Your machine tool builder defines in detail how the axes will move.

Please note while programming!

NOTICE

Danger of collision!

This cycle may perform extensive movements in one or more axes in rapid traverse! If you program the cycle parameter **Q570**=1, the feed rate and rapid traverse potentiometers, and, if applicable, the spindle potentiometer, have no effect. However, you can stop any movement by setting the feed rate potentiometer to zero. There is a danger of collision!

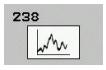
- Before recording measured data, test the cycle in test mode with **Q570=0**
- Contact your machine tool builder to learn about the type and range of movements in Cycle 238 before using the cycle.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Cycle 238 is CALL-active.

Ensure that the axes are not clamped before you start the measurement.

Cycle parameters



- **Q570 Mode (0=test/1=measure)?**: Here, you can specify whether the control will perform a measurement of the machine status in test mode or in measurement mode:
  0: No measured data are created. You can control the axis movements with the feed rate and rapid traverse potentiometers
  1: The cycle creates measured data. You **cannot** control the axis movements with the feed rate and rapid traverse potentiometers

Example

```
62 CYCL DEF 238  MEASURE MACHINE
   STATUS
   Q570=+0      ;MODE
```

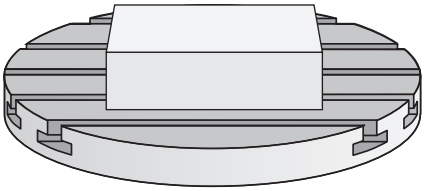
13.15 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143)

Cycle run

Refer to your machine manual!

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

Option 143 LAC (Load Adaptive Control) is required for Cycle 239.



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 the table axes. With option 143 LAC (Load Adaptive Control) and Cycle 239 **ASCERTAIN THE LOAD**, the control is able to automatically determine and adjust the actual mass inertia of the load, the actual friction forces and the maximum axis acceleration or reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The control performs a weighing procedure to ascertain the weight acting on the axes. With this weighing run, the axes move by a specified distance. Your machine tool builder defines the specific movements. 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.

In addition to adjusting the control parameters, with LAC the maximum acceleration is also adjusted in accordance with the weight. This enables the dynamics to be accordingly increased with low load to increase productivity.

Parameter Q570 = 0

1 There is no physical movement of the axes.
2 The control resets the LAC.
3 The control activates feedforward and, if applicable, controller parameters that allow safe movements of the axis/axes, independently of the current 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 control 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 control **depend** on the current load.
4 The control activates the ascertained parameters.

Please note while programming:

NOTICE

Danger of collision!

The cycle can execute extensive movements in several axes at rapid traverse!

- Ask your machine manufacturer about the type and extent of Cycle 239 before using it
- Before the cycle starts, the control moves to a safe position, if applicable. The machine tool builder determines this position.
- Set the potentiometers for feed-rate and rapid-traverse override to at least 50 % to ensure a correct ascertainment of the load.

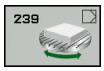
This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Cycle 239 becomes effective immediately after its definition.

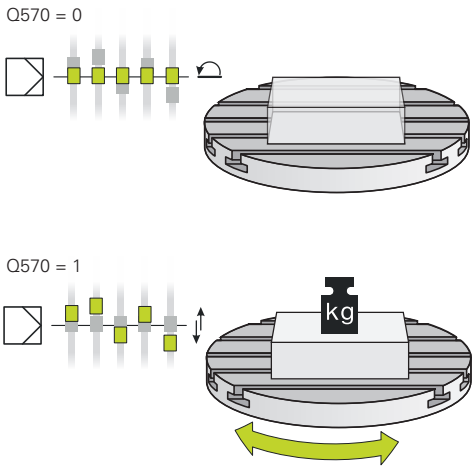
If you are using the mid-program startup function and the control thus skips Cycle 239 in the block scan, the control will ignore this cycle—no weighing procedure will be performed.

Cycle 239 supports the determination of the load on synchronized axes (gantry axes) if they have only one common position encoder (torque master slave).

Cycle parameters



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



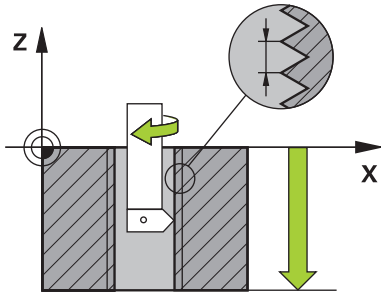
Example

```
62 CYCL DEF 239  ASCERTAIN THE LOAD
     Q570=+0     ;LOAD ASCERTATION
```

13.16 THREAD CUTTING (Cycle 18, DIN/ISO: G86)

Cycle run

Cycle **18** THREAD CUTTING moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.



Please note while programming:

NOTICE

Danger of collision!

A collision may occur if you do not program pre-positioning before calling Cycle 18. Cycle 18 does not perform approach and departure motion.

- Pre-position the tool before the start of the cycle.
- The tool moves from the current position to the entered depth after the cycle is called

NOTICE

Danger of collision!

If the spindle was switched on before calling the cycle, Cycle 18 switches the spindle off and the cycle works with a stationary spindle! Cycle 18 switches the spindle on again at the end if it was switched on before cycle start.

- Before starting this cycle, be sure to program a spindle stop! (For example with M5.)
- At the end of Cycle 18, the control restores the spindle to its state at cycle start. If the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle 18.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): SpindlePotentiometer (feed rate override is not active) and FeedPotentiometer (speed override is not active); the control then adjusts the spindle speed as required
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
- **limitSpindleSpeed** (no. 113604): Spindle speed limit
  True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time
  False: (Limiting not active)

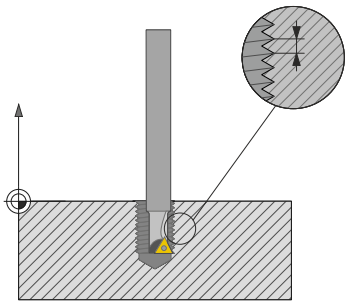
The spindle speed potentiometer is inactive.

Before calling this cycle, be sure to program a spindle stop! (For example with M5.) The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.

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

Cycle parameters

- Boring depth (incremental): Enter the thread depth based on the current position. Input range: –99999 to +99999
- Thread pitch: Enter the pitch of the thread. The algebraic sign entered here differentiates between right-hand and left-hand threads:
  + = right-hand thread (M3 with negative hole depth)
  – = left-hand thread (M4 with negative hole depth)



Example

```
25 CYCL DEF 18.0  THREAD CUTTING
26 CYCL DEF 18.1 DEPTH = -20
27 CYCL DEF 18.2 PITCH = +1
```

13.17 Programming examples

Example: Interpolation turning with Cycle 291

The following NC program illustrates the use of Cycle **291 COUPLG.TURNG.INTERP.** This programming example illustrates the machining of an axial recess and a radial recess.

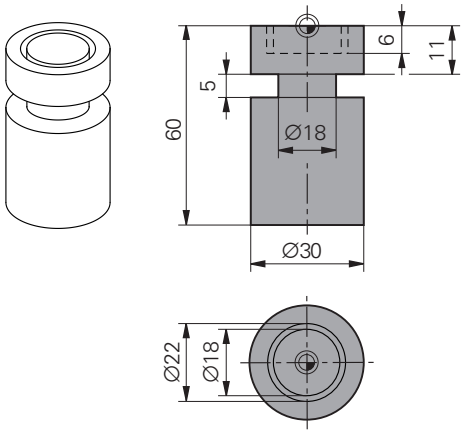
Tools

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

Program run

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

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



0 BEGIN PGM 1 MM	
1 BLK FORM CYLINDER Z R15 L60	Workpiece blank definition: cylinder
2 TOOL CALL 10	Tool call: tool for axial recess
3 CC X+0 Y+0	
4 LP PR+30 PA+0 R0 FMAX	Retract the tool
5 CYCL DEF 291 COUPLG.TURNG.INTERP.	Activate interpolation turning
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
Q561=+1 ;DREHWKZ. WANDELN	
6 CYCL CALL	Call the cycle
7 LP PR+9 PA+0 RR FMAX	Position the tool in the working plane
8 L Z+10 FMAX	
9 L Z+0.2 F2000	Position the tool in the spindle axis
10 LBL 1	Recessing on level surface, infeed: 0.2 mm, depth: 6 mm
11 CP IPA+360 IZ-0.2 DR+ F10000	
12 CALL LBL 1 REP 30	

13 LBL 2	Retract from recess, step: 0.4 mm
14 CP IPA+360 IZ+0.4 DR+	
15 CALL LBL 2 REP15	
16 L Z+200 R0 FMAX	Retract to clearance height, deactivate radius compensation
17 CYCL DEF 291 COUPLG.TURNG.INTERP.	Deactivate interpolation turning
Q560=+0 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
Q561=+0 ;DREHWKZ. WANDELN	
18 CYCL CALL	Call the cycle
19 TOOL CALL 11	Tool call: tool for radial recess
20 CC X+0 Y+0	
21 LP PR+25 PA+0 R0 FMAX	Retract the tool
22 CYCL DEF 291 COUPLG.TURNG.INTERP.	Activate interpolation turning
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
Q561=+1 ;DREHWKZ. WANDELN	
23 CYCL CALL	Call the cycle
24 LP PR+15.2 PA+0 RR FMAX	Position the tool in the working plane
25 L Z+10 FMAX	
26 L Z-11 F7000	Position the tool in the spindle axis
27 LBL 3	Recessing on lateral surface, infeed: 0.2 mm, depth: 6 mm
28 CC X+0.1 Y+0	
29 CP IPA+180 DR+ F10000	
30 CC X-0.1 Y+0	
31 CP IPA+180 DR+	
32 CALL LBL 3 REP15	
33 LBL 4	Retract from recess, step: 0.4 mm
34 CC X-0.2 Y+0	
35 CP IPA+180 DR+	
36 CC X+0.2 Y+0	
37 CP IPA+180 DR+	
38 CALL LBL 4 REP8	
39 LP PR+50 FMAX	
40 L Z+200 R0 FMAX	Retract to clearance height, deactivate radius compensation
41 CYCL DEF 291 COUPLG.TURNG.INTERP.	Deactivate interpolation turning
Q560=+0 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	

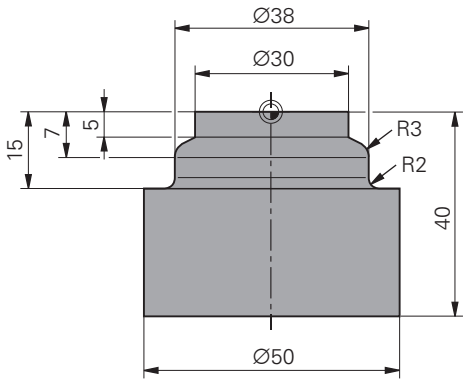
Q561=+0 ;DREHWKZ. WANDELN	
42 CYCL CALL	Call the cycle
43 TOOL CALL 11	Repeated **TOOL CALL** in order to reset the conversion of parameter Q561
44 M30	
45 END PGM 1 MM	

Example: Interpolation Turning Cycle 292

The following NC program illustrates the use of Cycle **292 CONTOUR.TURNG.INTRP.** This example illustrates the machining of an outside contour with the milling spindle rotating.

Program run

- Tool call: Milling cutter D20
- Cycle 32 Tolerance
- Reference to the contour with Cycle 14
- Cycle 292 Contour turning interpolation



0 BEGIN PGM 2 MM	
1 BLK FORM CYLINDER Z R25 L40	Workpiece blank definition: cylinder
2 TOOL CALL "D20" Z S111	Tool call: End mill D20
3 CYCL DEF 32.0 TOLERANCE	Use Cycle 32 to define the tolerance
4 CYCL DEF 32.1 T0.05	
5 CYCL DEF 32.2 HSC-MODE:1	
6 CYCL DEF 14.0 CONTOUR	Use Cycle 14 to refer to the contour in LBL1
7 CYCL DEF 14.1 CONTOUR LABEL1	
8 CYCL DEF 292 CONTOUR.TURNG.INTRP.	Define Cycle 292
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q546=+3 ;CHANGE TOOL DIRECTN.	
Q529=+0 ;MACHINING OPERATION	
Q221=+0 ;SURFACE OVERSIZE	
Q441=+1 ;INFEED	
Q449=+15000 ;FEED RATE	
Q491=+15 ;CONTOUR START RADIUS	
Q357=+2 ;CLEARANCE TO SIDE	
Q445=+50 ;CLEARANCE HEIGHT	
9 L Z+50 R0 FMAX M3	Pre-position in the tool axis, spindle on
10 L X+0 Y+0 R0 FMAX M99	Pre-position in the working plane to the center of rotation, call the cycle
11 LBL 1	LBL1 contains the contour
12 L Z+2 X+15	
13 L Z-5	
14 L Z-7 X+19	
15 RND R3	
16 L Z-15	
17 RND R2	
18 L X+27	

19 LBL 0	
20 M30	End of program
21 END PGM 2 MM	

Example of hob milling

The following NC program illustrates the use of Cycle 286 **GEAR HOBBING**. This programming example shows how to machine an involute spline with module=1 (deviating from DIN 3960).

Program run

- Tool call: Gear hob
- Start turning mode
- Reset the coordinate system with Cycle 801
- Move to safe position
- Define Cycle 285
- Call Cycle 286
- Reset the coordinate system with Cycle 801

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI+58	Workpiece blank definition: cylinder
2 TOOL CALL "ABWAELZFRAESER"	Call the tool
3 FUNCTION MODE TURN	Activate turning mode
4 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	Reset the coordinate system
5 M145	Deactivate M144 if still active
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
7 M140 MB MAX	Retract the tool
8 L A+0 R0 FMAX	Set the rotary axis to 0
9 L X0 Y0 R0 FMAX	Pre-position the tool in the workpiece center
10 Z+50 R0 FMAX	Pre-position the tool in the spindle axis
11 CYCL DEF 285 ZAHNRAD DEFINIEREN	Define Cycle 285
Q551=+0 ;STARTING POINT IN Z	
Q552=-11 ;END POINT IN Z	
Q540=+1 ;MODULE	
Q541=+90 ;NUMBER OF TEETH	
Q542=+90 ;OUTSIDE DIAMETER	
Q563=+1 ;TOOTH HEIGHT	
Q543=+0.05 ;TROUGH-TIP CLEARANCE	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 286 ZAHNRAD WAELZFRAESEN	Define Cycle 286
Q215=+0 ;MACHINING OPERATION	
Q200=+2 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q545=+1.6 ;TOOL LEAD ANGLE	
Q546=+0 ;CHANGE ROTATION DIR.	
Q547=+0 ;ANG. OFFSET, SPINDLE	
Q550=+1 ;MACHINING SIDE	
Q533=+1 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	

Q253=+2222 ;F PRE-POSITIONING	
Q553=+5 ;TOOL LENGTH OFFSET	
Q554=+10 ;SYNCHRONOUS SHIFT	
Q548=+1 ;ROUGHING SHIFT	
Q463=+1 ;MAX. CUTTING DEPTH	
Q488=+0.3 ;PLUNGING FEED RATE	
Q478=+0.3 ;PLUNGING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	
Q505=+0.2 ;FINISHING FEED RATE	
Q549=+3 ;FINISHING SHIFT	
13 CYCL CALL M303	Call the cycle, spindle ON
14 FUNCTION MODE MILL	Activate milling mode
15 M140 MB MAX	Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	Reset the rotation
17 M30	End of program
18 END PGM 5 MM	

Example of skiving

The following NC program illustrates the use of Cycle 287 **GEAR SKIVING**. This programming example shows how to machine an involute spline with module=1 (deviating from DIN 3960).

Program run

- Tool call: Internal gear cutter
- Start turning mode
- Reset the coordinate system with Cycle 801
- Move to safe position
- Define Cycle 285
- Call Cycle 287
- Reset the coordinate system with Cycle 801

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI+58	Workpiece blank definition: cylinder
2 TOOL CALL "Hohlradfraeser"	Call the tool
3 FUNCTION MODE TURN	Activate turning mode
4 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	Reset the coordinate system
5 M145	Deactivate M144 if still active
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
7 M140 MB MAX	Retract the tool
8 L A+0 R0 FMAX	Set the rotary axis to 0
9 L X0 Y0 R0 FMAX	Pre-position the tool in the workpiece center
10 Z+50 R0 FMAX	Pre-position the tool in the spindle axis
11 CYCL DEF 285 ZAHNRAD DEFINIEREN	Define Cycle 285
Q551=+0 ;STARTING POINT IN Z	
Q552=-11 ;END POINT IN Z	
Q540=+1 ;MODULE	
Q541=+90 ;NUMBER OF TEETH	
Q542=+90 ;OUTSIDE DIAMETER	
Q563=+1 ;TOOTH HEIGHT	
Q543=+0.05 ;TROUGH-TIP CLEARANCE	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 287 ZAHNRAD WAELZSCHAELEN	Define Cycle 287
Q240=+5 ;NUMBER OF CUTS	
Q584=+1 ;NO. OF FIRST CUT	
Q585=+5 ;NO. OF LAST CUT	
Q200=+2 ;SET-UP CLEARANCE	
Q260=+50 ;CLEARANCE HEIGHT	
Q545=+20 ;TOOL LEAD ANGLE	
Q546=+0 ;CHANGE ROTATION DIR.	
Q547=+0 ;ANG. OFFSET, SPINDLE	
Q550=+1 ;MACHINING SIDE	

Q533=+1 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	
Q253=+2222 ;F PRE-POSITIONING	
Q586=+0,4 ;FIRST INFEED	
Q587=+0,1 ;LAST INFEED	
Q588=+0,4 ;FIRST FEED RATE	
Q589=+0,25 ;LAST FEED RATE	
Q580=+0,2 ;FEED-RATE ADAPTION	
13 CYCL CALL M303	Call the cycle, spindle ON
14 FUNCTION MODE MILL	Activate milling mode
15 M140 MB MAX	Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	Reset the rotation
17 M30	End of program
18 END PGM 5 MM	

14

Cycles: Turning

14.1 Turning cycles (option 50)

Overview

Proceed as follows to define turning cycles:

CYCL DEF

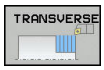
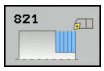
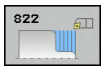
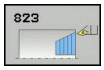
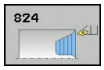
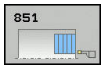
- Press the **CYCL DEF** key

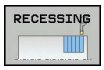
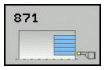
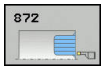
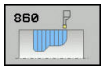
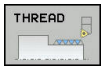
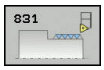
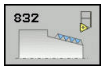
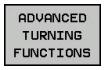
TURNING

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

The control offers the following cycles for turning operations:

Soft key	Cycle group	Cycle	Page
SPECIAL CYCLES	**Special cycles**		
800		ADJUST XZ SYSTEM (Cycle 800, DIN/ISO: G800)	432
801		RESET ROTARY COORDINATE SYSTEM (Cycle 801, DIN/ISO: G801)	440
880		GEAR HOBBING (Cycle 880, DIN/ISO: G880, option 131)	442
892		CHECK UNBALANCE (Cycle 892, DIN/ ISO: G892)	450
LONGITDNL	**Cycles for longitudinal turning**		453
811		TURN SHOULDER, LONGITUDINAL(Cycle 811, DIN/ ISO: G812)	454
812		TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, ISO: G812)	457
813		TURN PLUNGE CONTOUR LONGITUDINAL (Cycle 813, DIN/ISO: G813)	460
814		TURN PLUNGE LONGITUDINAL EXTENDED (Cycle 814, DIN/ISO: G814)	463
810		TURN CONTOUR LONGITUDINAL(Cycle 810, DIN/ISO: G812)	466
815		CONTOUR-PARALLEL TURNING (Cycle 815, DIN/ISO: G815)	470

Soft key	Cycle group	Cycle	Page
	**Cycles for transverse turning**		453
		TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)	473
		TURN SHOULDER FACE EXTENDED (Cycle 822, DIN/ISO: G822)	476
		TURN TRAVERSE PLUNGE (Cycle 823, DIN/ISO: G823)	480
		TURN PLUNGE TRANSVERSE EXTENDED (Cycle 824, DIN/ISO: G824)	483
		TURN CONTOUR TRANSVERSE (Cycle 820, DIN/ISO: G820)	487
	Cycles for recess turning		
		SIMPLE REC. TURNG., RADIAL DIR. (Cycle 841, DIN/ISO: G841)	491
		EXPANDED RECESS TURNING, RADIAL (Cycle 842, DIN/ISO: G842)	494
		SIMPLE RECESS TURNING, AXIAL (Cycle 851, DIN/ISO: G851)	498
		ENHANCED RECESS TURNING, AXIAL (Cycle 852, DIN/ISO: G852)	501
		CONTOUR RECESS TURNING, RADIAL (Cycle 840, DIN/ISO: G840)	505
		CONTOUR RECESS TURNING, AXIAL (Cycle 850, DIN/ISO: G850)	509

Soft key	Cycle group	Cycle	Page
	**Cycles for recessing**		
		SIMPLE RECESSING, RADIAL (Cycle 861, DIN/ISO: G861)	513
		EXPANDED RECESSING, RADIAL (Cycle 862, DIN/ISO: G862)	516
		SIMPLE RECESSING, AXIAL (Cycle 871, DIN/ISO: G871)	520
		EXPANDED RECESSING, AXIAL (Cycle 872, DIN/ISO: G872)	523
		CONTOUR RECESSING, RADIAL (Cycle 860, DIN/ISO: G860)	528
		CONTOUR RECESSING, AXIAL (Cycle 870, DIN/ISO: G870)	532
	**Cycles for thread turning**		
		THREAD, LONGITUDINAL (Cycle 831, DIN/ISO: G831)	536
		THREAD, EXTENDED (Cycle 832, DIN/ISO: G832)	540
		THREAD, CONTOUR-PARALLEL (Cycle 830, DIN/ISO: G830)	544
	**Advanced turning functions**		
		TURNING, SIMULTANEOUS FINISHING (Cycle 883, DIN/ISO: G883, (option 158)	548

Working with turning cycles

In turning cycles, the control takes the cutting geometry (**TO, RS, P-ANGLE, T-ANGLE**) of the tool into account in order to prevent damage to the defined contour elements. If it is not possible to machine the entire contour with the active tool, the control will display a warning.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the control detects the machining position (inside or 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 the direction of rotation.

If you program **M136** before a cycle, the control interprets feed rate values in the cycle in mm/rev.; without **M136** in mm/min.

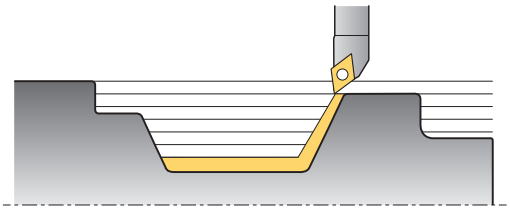
If you execute turning cycles with inclined machining (**M144**), the angles of the tool with respect to the contour change. The control automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

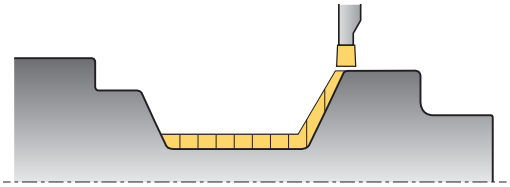
Some cycles machine contours that you have written in a subprogram. You program these contours with path functions or FK functions. Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

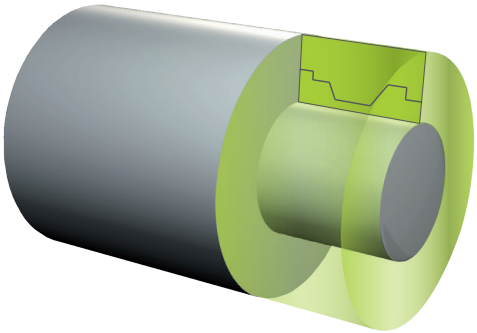
You must call turning cycles 880 and 81x - 87x with **CYCL CALL** or **M99**. Before calling a cycle, be sure to program:

- Turning mode: **FUNCTION MODE TURN**
- Call a tool with **TOOL CALL**
- Direction of rotation of turning spindle, e.g. **M303**
- Selection of speed or cutting speed: **FUNCTION TURNDATA SPIN**
- If you use feed rate per revolution mm/rev., **M136**
- Position the tool to a suitable starting point e.g. **L X+130 Y +0 R0 FMAX**
- Adapt the coordinate system and align the tool: **CYCL DEF 800 ADJUST XZ SYSTEM**

Blank form update (FUNCTION TURNDATA)







During turning operations workpieces must often be machined with several tools. Often a contour element cannot be completely finished because the tool shape does not permit this (e.g. with an undercut). In this case, single sub-areas have to be reworked with other tools. Using the contour follow-up function, the control detects the already machined areas and adapts all approach and departure paths to the specific, current machining situation. With the shorter machining paths, air cuts are avoided to significantly reduce machining time.

To activate contour follow-up, program the **TURNDATA BLANK** function and reference an NC program or subprogram with a workpiece blank specification. The workpiece blank defined in **TURNDATA BLANK** determines the area to be machined with the contour follow-up. **TURNDATA BLANK OFF** deactivates the contour follow-up.

NOTICE

Danger of collision!

Contour follow-up is used to optimize machining areas and approach movements. For approach and departure paths, the control takes the specific workpiece blank into account that is being followed. If parts of the finished part extend beyond the workpiece blank, this may damage the workpiece and tool.

- Define the workpiece blank larger than the finished part

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 control provides various options for defining the workpiece blank:

Soft key	Workpiece blank definition
BLANK OFF	Deactivate blank form update **TURNDATA BLANK OFF**: No input
BLANK <FILE>	Workpiece blank definition in an NC program: Enter the name of the file
BLANK <FILE>=QS	Workpiece blank definition in an NC program: Enter the string parameter with the program name
BLANK LBL NR	Workpiece blank definition in a subprogram: Enter the number of the subprogram
BLANK LBL NAME	Workpiece blank definition in a subprogram: Enter the name of the subprogram

Soft key	Workpiece blank definition
BLANK LBL QS	Workpiece blank definition in a subprogram: Enter the string parameter with the subprogram name

Proceed as follows to activate blank form update and define the workpiece blank:

SPEC FCT

- Press the **SPEC FCT** key

TURNING PROGRAM FUNCTIONS

- Press the **TURNING PROGRAM FUNCTIONS** soft key

FUNCTION TURNDATA

- Press the **FUNCTION TURNDATA** soft key

TURNDATA BLANK

- Press the **TURNDATA BLANK** soft key

Example

```
11 FUNCTION TURNDATABLANK LBL 20
```

14.2 ADJUST XZ SYSTEM (Cycle 800, DIN/ISO: G800)

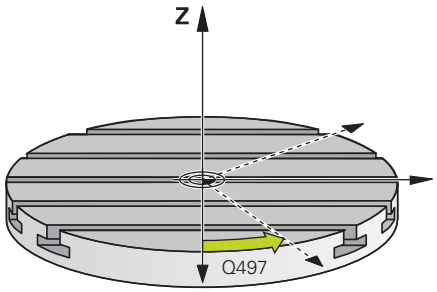
Application

Refer to your machine manual!

Option 50 must have been enabled.

Option 135 must have been enabled.

This function must be adapted by your machine manufacturer.



In order to be able to perform a turning operation, you need to position the tool appropriately with respect to the turning spindle. You can use Cycle **800 ADJUST XZ SYSTEM** for this purpose.

With turning operations, the inclination angle between the tool and turning spindle is important, in order to e.g. machine contours with undercuts. Cycle 800 provides various possibilities for aligning the coordinate system for an inclined machining operation:

- If you have positioned the tilting axis for inclined machining, you can use Cycle 800 to orient the coordinate system to the positions of the tilting axes (**Q530=0**). In this case, make sure to program **M144** or **M128/TCPM** for proper calculation of the orientation
- Cycle 800 calculates the required angle of the tilting axis based on the inclination angle **Q531**. Depending on the strategy selected in parameter INCLINED MACHINING **Q530**, the control positions the tilting axis with (**Q530=1**) or without compensation movement (**Q530=2**)
- Cycle 800 uses the inclination angle **Q531** to calculate the required tilting axis angle, but does not position the tilting axis (**Q530=3**). You need to position the tilting axis manually to the calculated values **Q120** (A axis), **Q121** (B axis), and **Q530** (C axis) after the cycle

If you change the position of a tilting axis, you need to run Cycle 800 again to align the coordinate system.

If the milling spindle axis and the turning spindle axis are parallel to each other, you can use the **Precession angle Q497** to define any desired rotation of the coordinate system about 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 turning spindle and milling spindle axes are not parallel, only two precession angles are realistic for machining. The control selects the angle that is closest to the input value of **Q497**.

Cycle 800 positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can use a mirrored version of the tool (**REVERSE TOOL Q498**); this offsets the milling spindle by 180°. In this way, you can use your tools both for inside and outside machining. Position the cutting edge at the center of the turning spindle using a positioning block, such as **L Y+0 R0 FMAX**.

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of rotation is aligned with the axis of the turning spindle. For example, this is the case with large or rotationally non-symmetric workpieces. The eccentric turning **Q535** function in Cycle 800 enables you to perform turning such cases as well.

During eccentric turning, more than one linear axis is coupled to the turning spindle. The control compensates the eccentricity by performing circular compensating movements with the coupled linear axes.

This function must be enabled and adapted by the machine tool builder.

If you machine with high speed and a high amount of eccentricity, you need to program large feed rates for the linear axes in order to perform the movements synchronously. If these feed rates are not met, the contour would be damaged. The control therefore generates an error message if 80 % of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the speed.

NOTICE

Danger of collision!

The Control performs compensating movements during coupling and decoupling. Check for possible collisions.

- Coupling and decoupling must be performed while the spindle is stationary.

NOTICE

Danger of collision!

Collision monitoring (DCM) is not active during eccentric turning. The control displays a corresponding warning during eccentric turning.

- Watch out for possible collisions!

NOTICE

Danger of collision!

The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.

- Select the specifications in such a way that no vibrations (resonances) occur.

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 control only in the ACTUAL value position display.

Effect

With Cycle 800 **ADJUST XZ SYSTEM**, the control 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 implicitly reset by other factors:

- Mirroring of tool data (**Q498 REVERSE TOOL**) is reset by a tool call with **TOOL CALL**
- The **ECCENTRIC TURNING Q535** function is reset at the end of the program or if the program is aborted (internal stop)

Please note while programming:

NOTICE

Danger of collision!

If the milling spindle was defined as an NC axis in turning mode, the control is able to derive a tool reversal from the axis position. However, if the milling spindle was defined as a spindle, there is a risk that the tool reversal definition might get lost!
In both cases, proceed as follows:

- Enable tool reversal again after a **TOOL CALL** block

NOTICE

Danger of collision!

If **Q498**=1 and you additionally program the **FUNCTION LIFTOFF ANGLE TCS** function, there might be different results, depending on the configuration. If the tool spindle has been defined as an axis, the **LIFTOFF** will be included in the rotation when reversing the tool. If the tool spindle has been defined as a kinematic transformation, the **LIFTOFF** will **not** be included in the rotation when reversing the tool!

- Carefully test the NC program or program section in **Program run, single block** operating mode
- If required, change the algebraic sign of the SPB angle.

Cycle 800 **ADJUST XZ SYSTEM** is machine-dependent. Refer to your machine manual!

The machine tool builder determines the configuration of your machine. In this configuration if the tool spindle is defined as the axis in the kinematics, the feed-rate potentiometer is effective with movements with Cycle 800.

The machine tool builder can determine how exactly the precession angle aligns the tool.

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

The tool must be clamped and measured in the correct position.

You can mirror the tool data (**Q498 REVERSE TOOL**) only if a turning tool has been selected.

Check the orientation of the tool before machining.

To reset Cycle 800, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM**.

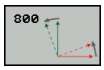
Cycle 800 limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You may have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle 800. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle 800, the smaller value will be applied. To reset Cycle 800, program Cycle 801. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** applies again.

Cycle 800 will only position the first rotary axis based on the tool position. If you want to move other rotary axes to a specific position, position these axis correspondingly before running Cycle 800.

If you set parameter **Q530** Inclined machining to 0 (tilting axes must have been positioned previously), make sure to program M144 or TCPM/M128 beforehand.

If you use the 1: MOVE, 2: TURN, and 3: STAY settings in parameter **Q530** Inclined machining, the control will activate (depending on the machine configuration) the **M144** function or TCPM. (**Further details:** User's Manual for Setup, Testing and Running NC programs.)

Cycle parameters



- **Q497 Precession angle?**: Angle to which the control aligns the tool. Input range: 0 to 359.9999
- **Q498 Reverse tool (0=no/1=yes)?**: Mirror tool for inside/outside machining. Input range: 0 or 1.
- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  0: Maintain position of tilted axes (axis must already have been positioned)
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
  3: Do not position the tilting axis. Position the tilting axes later in a separate positioning block (STAY). The control stores the position values in the parameters **Q120** (A axis), **Q121** (B axis) and **Q122** (C axis).
- **Q531 Angle of incidence?**: Angle of incidence for aligning the tool. Input range: –180.000° to +180.000°
- **Q532 Feed rate for positioning?**: Traverse speed of the tilting axis during automatic positioning. Input range: 0.001 to 99999.999

- **Q533 Preferred dir. of incid. angle?**: Selection of alternative inclination options. The angle of incidence you define is used by the control 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 control should apply:
  0: Option with the shortest distance from the current position
  –1: Option in the range between 0° and –179.9999°
  +1: Option in the range between 0° and +180°
  –2: Option in the range between –90° and –179.9999°
  +2: Option in the range between +90° and +180°
- **Q535 Eccentric turning?**: Couple the axes for the eccentric turning operation:
  0: Deactivate axis couplings
  1: Activate axis couplings. The center of rotation is located at the active preset
  2: Activate axis couplings. The center of rotation is located at the active datum
  3: Do not change axis couplings
- **Q536 Eccentric turning without stop?**: Interrupt program run before the axes are coupled:
  0: Stop before the axes are coupled again. In stopped condition, the control opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. Then press **NC start** to resume machining or press the **CANCEL** soft key to cancel machining
  1: Axes are coupled without stopping beforehand

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

Please note while programming:

Cycle **RESET ROTARY COORDINATE SYSTEM** is machine-dependent. Refer to your machine manual!

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

With Cycle 801 **RESET ROTARY COORDINATE SYSTEM**, you can reset the settings you have made with Cycle 800 **ADJUST XZ SYSTEM**.

To reset Cycle 800, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM**.

Cycle 800 limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You may have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle 800. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle 800, the smaller value will be applied. To reset Cycle 800, program Cycle 801. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** applies again.

Effect

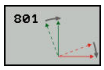
Cycle 801 resets the following settings you have programmed with Cycle 800:

- Precession angle **Q497**
- Reverse tool **Q498**

If you have executed the eccentric turning function with Cycle 800, please note the following: Cycle 800 limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You may have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle 800. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle 800, the smaller value will be applied. To reset Cycle 800, program Cycle 801. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** applies again.

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.

14.4 GEAR HOBBING (Cycle 880, DIN/ISO: G880, option 131)

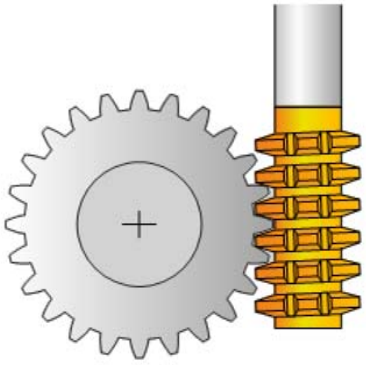
Cycle run

Refer to your machine manual!

Option 50 must have been enabled.

Option 131 must have been enabled.

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



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 control positions the tool in the tool axis to clearance height **Q260** at the feed rate FMAX. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.

2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the FMAX feed rate. 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 control then tilts the working plane at the feed rate **Q253**; **M144** is internally active in the cycle

4 The control positions the tool at the feed rate FMAX to the starting point in the working plane.

5 The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q460**.

6 The control now moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. 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 control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point

8 The control repeats the steps 5 to 7 until the defined gear is completed.

9 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX

10 The machining operation ends in the tilted system.

11 Now you need to move the tool to a safe height and reset the tilting of the working plane.
12 Then you must program Cycle 801 RESET ROTARY COORDINATE SYSTEM and **M145**.

Please note while programming:

NOTICE

Danger of collision!

If you do not position the tool to a safe position, a collision may occur between the tool and workpiece (fixtures) during tilting.

- Pre-position the tool so that it is already on the desired machining side **Q550**.
- Move the tool to a safe position on this machining side

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q460**!

- Clamp the workpiece out of the fixtures far enough to prevent a danger of collision between the tool and the fixtures
- Clamp the workpiece in such a way that its protrusion from the fixture will not cause any collision when the tool is automatically moved to the starting or end point using a path that is extended by the set-up clearance **Q460**

NOTICE

Danger of collision!

Depending on whether you use M136 or not, the feed rate values will be interpreted differently by the control. If the programmed feed rate was too high, the workpiece might be damaged.

- If you program M136 explicitly before the cycle, the control will interpret the feed rates in the cycle in mm/rev.
- If you do not program M136 before the cycle, the control will interpret the feed rates in the cycle in mm/min.

NOTICE

Danger of collision!

If you do not reset the coordinate system after Cycle 880, the precession angle set by the cycle will remain in effect.

- Make sure to program Cycle 801 after Cycle 800 in order to reset the coordinate system.
- Make sure to program Cycle 801 after a program abort in order to reset the coordinate system.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

The cycle is CALL-active.

The values entered for module, number of teeth and outside diameter are monitored. If these values are not coherent, an error message displays. You can fill in 2 of the 3 parameters. Enter 0 for the module, the number of teeth, or the outside diameter. In this case, the control will calculate 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.

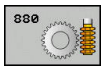
Define the tool as a milling cutter in the tool table.

In order to avoid that the maximum permissible spindle speed of the tool is not exceeded, you can program a limitation. (Specify it in the **Nmax** column of the "tool.t" tool table.)

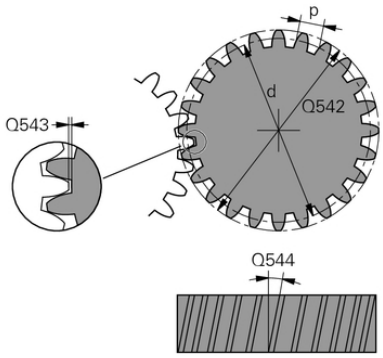
Before starting the cycle, program the direction of rotation of the workpiece (M303/M304).

Before cycle call, set the preset to the center of rotation.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q540 Module?**: Define the gear: Module of the gear wheel. Input range: 0 to 99.9999
- **Q541 Number of teeth?**: Define the gear: Number of teeth. Input range: 0 to 99999
- **Q542 Outside diameter?**: Define the gear: Outside diameter of the finished part. Input range: 0 to 99999.9999
- **Q543 Trough-to-tip clearance?**: Define the gear: Distance between the tip circle of the gear to be cut and the root circle of the mating gear. Input range: 0 to 9.9999
- **Q544 Angle of inclination?**: Define the gear: Angle by which teeth of a helical gear are inclined relative to the direction of the axis (For straight-cut gears this angle is 0°.) Input range: -60 to +60
- **Q545 Tool lead angle?**: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range: –60.0000 to +60.0000
- **Q546 Reverse tool rotation direction?**: Define the tool: Direction of spindle rotation of the gear hob:
  3: Tool turns clockwise (M3)
  4: Tool turns counterclockwise (M4)
- **Q547 Angle offset of tool spindle?**: Angle by which the control rotates the workpiece at the beginning of the cycle. Input range -180.0000 to +180.0000
- **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
  0: Positive machining side of the principal axis in the I-CS
  1: Negative machining side of the principal axis in the I-CS



Example

```
63 CYCL DEF 880  GEAR HOBBING
    Q215=0       ;MACHINING OPERATION
    Q540=0       ;MODULE
    Q541=0       ;NUMBER OF TEETH
    Q542=0       ;OUTSIDE DIAMETER
    Q543=0.167;TROUGH-TIP CLEARANCE
    Q544=0       ;ANGLE OF INCLINATION
    Q545=0       ;TOOL LEAD ANGLE
    Q546=3       ;CHANGE TOOL DIRECTN.
    Q547=0       ;ANG. OFFSET, SPINDLE
    Q550=1       ;MACHINING SIDE
    Q533=0       ;PREFERRED DIRECTION
    Q530=2       ;INCLINED MACHINING
    Q253=750     ;F PRE-POSITIONING
    Q260=100     ;CLEARANCE HEIGHT
    Q553=10      ;TOOL LENGTH OFFSET
    Q551=0       ;STARTING POINT IN Z
    Q552=-10     ;END POINT IN Z
    Q463=1       ;MAX. CUTTING DEPTH
    Q460=2       ;SAFETY CLEARANCE
    Q488=0.3     ;PLUNGING FEED RATE
    Q478=0.3     ;ROUGHING FEED RATE
    Q483=0.4     ;OVERSIZE FOR DIAMETER
    Q505=0.2     ;FINISHING FEED RATE
```

- **Q533 Preferred dir. of incid. angle?**: Selection of alternative inclination options. The inclination angle you define is used by the control to calculate the appropriate position of the tilting axes on your machine. In general, there are always two possible solutions. Using parameter **Q533**, configure which of the solution options the control should apply:
  0: Option with the shortest distance from the current position
  –1: Option in the range between 0° and –179.9999°
  +1: Option in the range between 0° and +180°
  –2: Option in the range between –90° and –179.9999°
  +2: Option in the range between +90° and +180°
- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control carries out a compensation movement with the linear axes
  2: Automatically position the tilting axis without orienting the tool tip (TURN)
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Entry in mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q260 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: –99999.9999 to 99999.9999
- **Q553 TOOL:L offset, machining start?** (incremental): Define the length offset (L OFFSET) from which the tool will machine. The tool will be offset in the longitudinal direction by this value. Input range: 0 to 999.9999

- **Q551 Starting point in Z?**: Starting point in Z for gear hobbing. Input range: -99999.9999 to 99999.9999
- **Q552 End point in Z?**: End point in Z for gear hobbing. Input range: -99999.9999 to 99999.9999
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning. Input range: 0 to 999.999
- **Q488 Feed rate for plunging**: Feed rate for tool infeed. Input range: 0 to 99999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control 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:

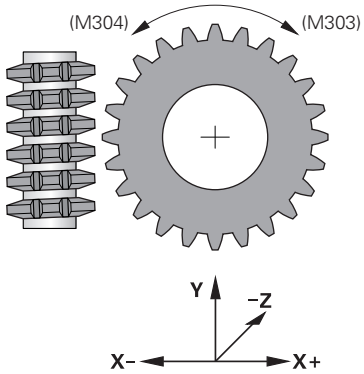
1 **What tool? (Right-cutting/left-cutting?)**

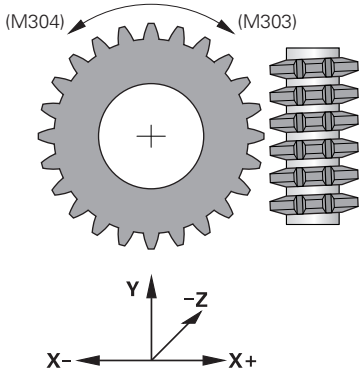
2 **What machining side? X+ (Q550=0) / X- (Q550=1)**

3 **Look up the direction of rotation of the rotary table in one of the two tables below!** To do so, select the appropriate table for the direction of rotation of your tool (**right-cutting/left-cutting**). Please refer to the tables below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)** ab.

Tool: Right-cutting M3	
Machining side X+ (Q550=0)	Direction of rotation of the table: Clockwise (M303)
Machining side X- (Q550=1)	Direction of rotation of the table: Counterclockwise (M304)

Tool: Left-cutting M4	
Machining side X+ (Q550=0)	Direction of rotation of the table: Counterclockwise (M304)
Machining side X- (Q550=1)	Direction of rotation of the table: Clockwise (M303)

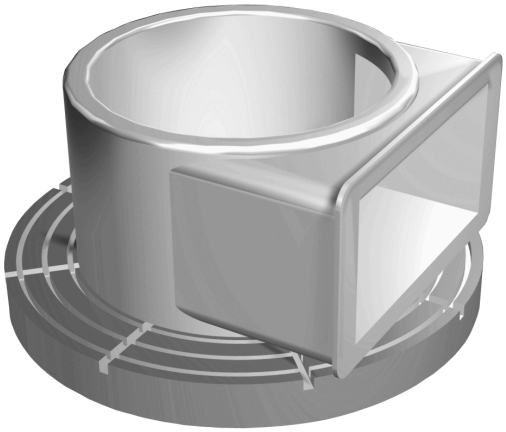




14.5 CHECK UNBALANCE (Cycle 892, DIN/ ISO: G892)

Application

Refer to your machine manual!
Option 50 must have been enabled.
Machine and control must be specially prepared by the machine tool builder for use of this cycle.



An unbalance can occur when turning an unsymmetrical workpiece, such as a pump body. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle **892 CHECK UNBALANCE**, the control checks the unbalance of the turning spindle. This cycle uses two parameters. **Q450** describes the maximum unbalance and **Q451** the maximum spindle speed. **If the maximum unbalance is exceeded, an error message is displayed and the NC program is aborted.** If the maximum unbalance is not exceeded, the control executes the NC program without interruption. This function protects the machine mechanics. It enables you to take action if an excessive unbalance is detected.

Please note while programming!

NOTICE

Danger of collision!

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance. If high unbalance loads are not compensated, this can lead to defects on the machine.

- Before starting a new machining cycle, run Cycle 892.
- If required, use balancing weights to compensate any unbalance.

NOTICE

Danger of collision!

The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps. If high unbalance loads are not compensated, this can lead to defects on the machine

- Make sure to run Cycle 892 between the machining steps.
- If required, use balancing weights to compensate any unbalance.

NOTICE

Danger of collision!

High unbalance loads, especially in combination with a high mass, might damage the machine. Keep in mind the mass and unbalance of the workpiece when choosing the speed.

- Do not program high speeds with heavy workpieces or high unbalance loads.

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.

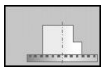
This function can also be run on machines with more than one turning spindle. Contact the machine tool builder for further information.

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.

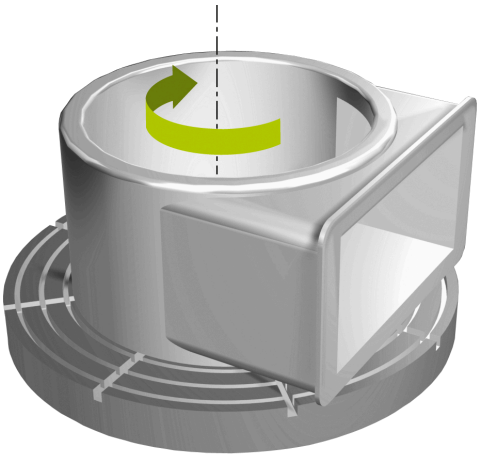
This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

If Cycle 892 CHECK UNBALANCE has aborted the NC program, it is recommended to use the manual MEASURE UNBALANCE cycle. With this cycle, the control determines the unbalance and calculates the mass and position of a balancing weight.
Further information: User's Manual for Setup, Testing and Running NC programs

Cycle parameters



- **Q450 Max. permissible runout?** Specifies the maximum runout of a sinusoidal unbalance signal in millimeters (mm). The signal results from the following error of the measuring axis and from the spindle revolutions.
- **Q451 Rotational speed?** Enter the rotational speed in revolutions per minute. The test for an unbalance begins with a low initial speed (e.g. 50 rpm). It is then automatically increased by specified increments (e.g. 25 rpm) until the maximum speed defined in parameter **Q451** is reached. Spindle speed override is disabled.



Example

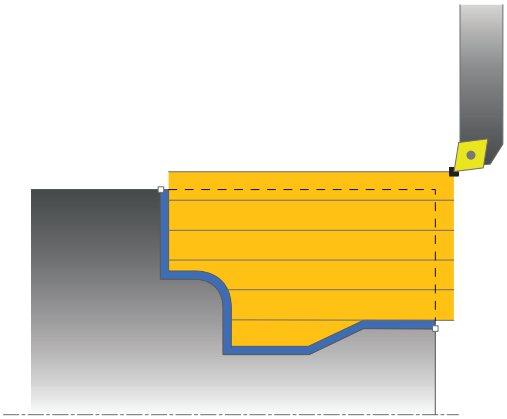
```
63 CYCL DEF 892 CHECK UNBALANCE
    Q450=0      ;MAXIMUM RUNOUT
    Q451=50     ;SPEED
```

14.6 Fundamentals of Turning Cycles

Refer to your machine manual!

Option 50 must have been enabled.

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



The pre-positioning of the tool has a decisive influence on the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when the cycle is called. When calculating the area to be machined, the control takes into account the starting point and the end point defined in the cycle or of contour defined in the cycle. If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand in some cycles.

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 Control takes the information for this from the position of the tool or the definition of the cycle (see "Working with turning cycles", Page 429).

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.

NOTICE

Danger of collision!

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.

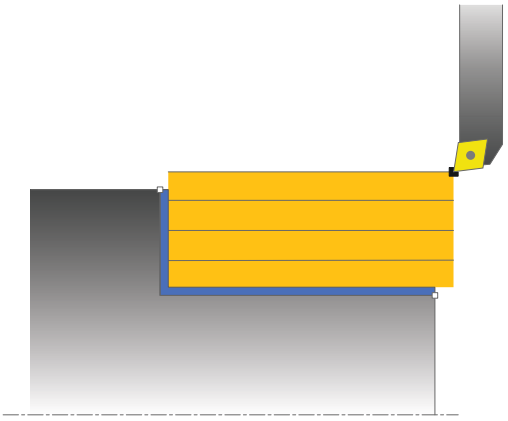
14.7 TURN SHOULDER, LONGITUDINAL (Cycle 811, DIN/ISO: G812)

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 control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 The control performs a paraxial infeed movement at rapid traverse.
3 The control finishes the contour of the finished part at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

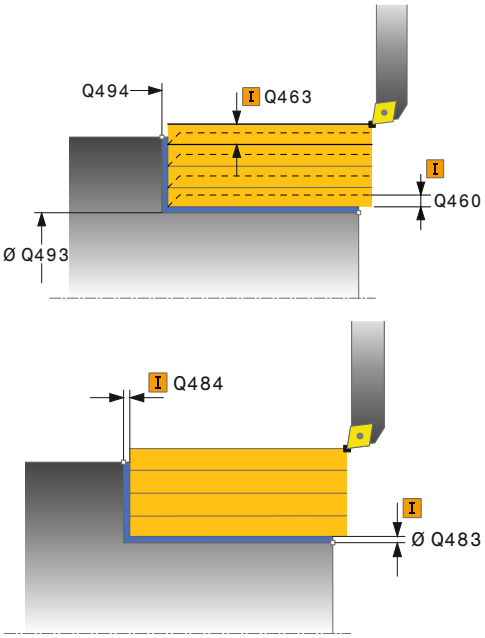
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

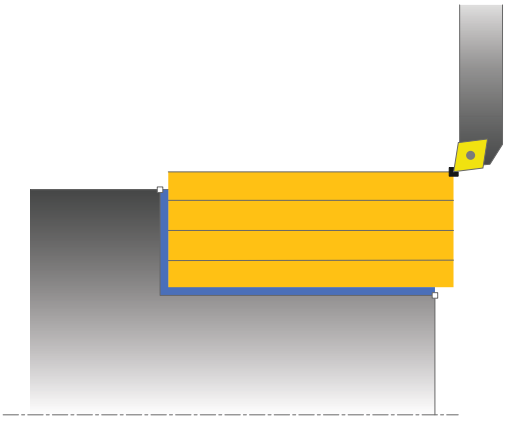


Example

```
11 CYCL DEF 811  SHOULDER,
   LONGITDNL.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q493=+50     ;DIAMETER AT CONTOUR
                 END
   Q494=-55     ;CONTOUR END IN Z
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.8 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, ISO: G812)

Application



This cycle enables you to run longitudinal turning of shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and starts the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand.

1 The control performs a paraxial infeed movement at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

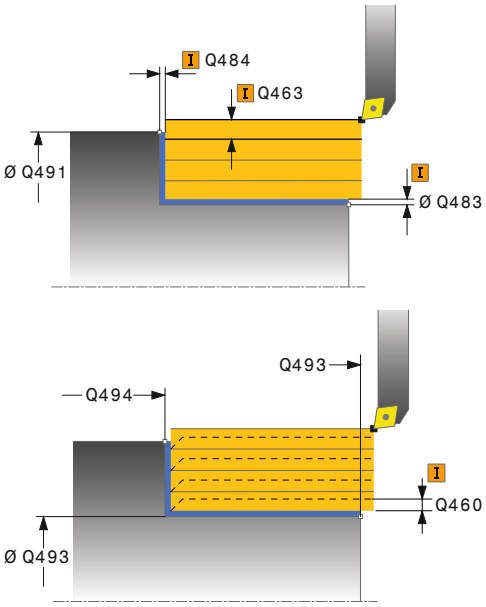
The tool position at cycle call (cycle start point) influences the area to be machined.

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of circumferen. surface?**: Angle between the circumferential surface and the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of face?**: Angle between the level surface and the rotary axis



Example

```
11 CYCL DEF 812  SHOULDER, LONG. EXT.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR START
   Q492=+0      ;CONTOUR START IN Z
   Q493=+50     ;DIAMETER AT CONTOUR END
   Q494=-55     ;CONTOUR END IN Z
```

- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

```
Q495=+5      ;ANGLE OF CIRCUM. SURFACE
Q501=+1      ;TYPE OF STARTING ELEMENT
Q502=+0.5    ;SIZE OF STARTING ELEMENT
Q500=+1.5    ;RADIUS OF CONTOUR EDGE
Q496=+0      ;ANGLE OF FACE
Q503=+1      ;TYPE OF END ELEMENT
Q504=+0.5    ;SIZE OF END ELEMENT
Q463=+3      ;MAX. CUTTING DEPTH
Q478=+0.3    ;ROUGHING FEED RATE
Q483=+0.4    ;OVERSIZE FOR DIAMETER
Q484=+0.2    ;OVERSIZE IN Z
Q505=+0.2    ;FINISHING FEED RATE
Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX M303
13 CYCL CALL
```

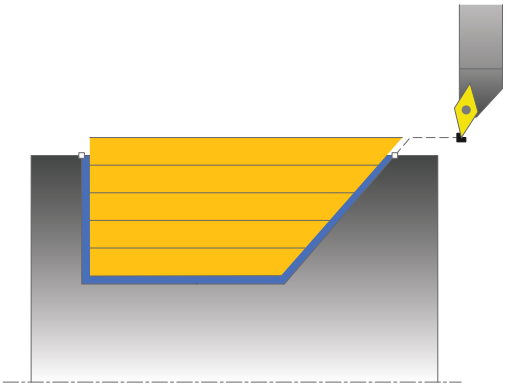
14.9 TURN PLUNGE CONTOUR LONGITUDINAL (Cycle 813, DIN/ISO: G813)

Application

This cycle enables you to run longitudinal turning of shoulders with plunging 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 control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The tool position at cycle call (cycle start point) influences the area to be machined.

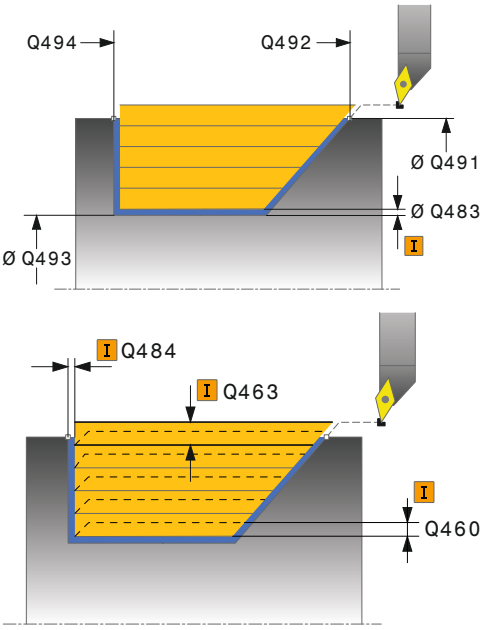
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging edge. This angle references a line perpendicular to the rotary axis.
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

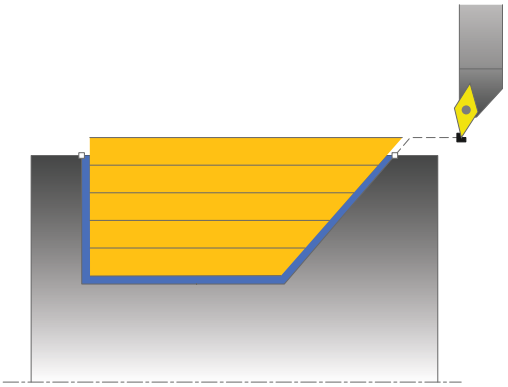


Example

```
11 CYCL DEF 813  TURN PLUNGE
   CONTOUR LONGITUDINAL
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR
                 START
   Q492=-10     ;CONTOUR START IN Z
   Q493=+50     ;DIAMETER AT CONTOUR
                 END
   Q494=-55     ;CONTOUR END IN Z
   Q495=+70     ;ANGLE OF SIDE
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

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

Application



This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The tool position at cycle call (cycle start point) influences the area to be machined.

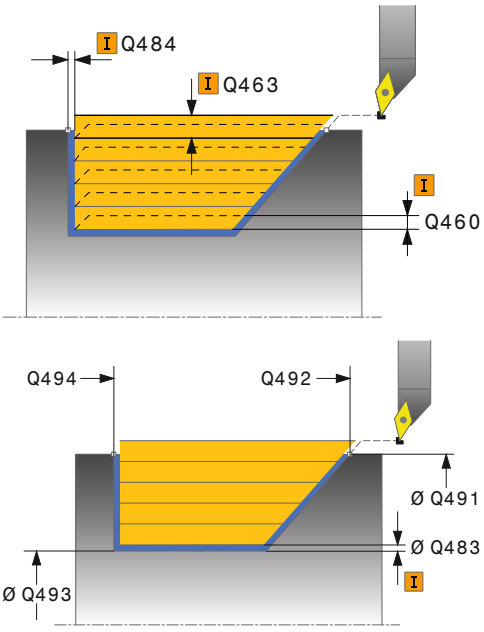
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging edge. This angle references a line perpendicular to the rotary axis.
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of face?**: Angle between the level surface and the rotary axis



Example

```
11 CYCL DEF 814  TURN PLUNGE LONGITUDINAL EXT.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR START
   Q492=-10     ;CONTOUR START IN Z
   Q493=+50     ;DIAMETER AT CONTOUR END
   Q494=-55     ;CONTOUR END IN Z
```

- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

```
Q495=+70    ;ANGLE OF SIDE
Q501=+1     ;TYPE OF STARTING ELEMENT
Q502=+0.5   ;SIZE OF STARTING ELEMENT
Q500=+1.5   ;RADIUS OF CONTOUR EDGE
Q496=+0     ;ANGLE OF FACE
Q503=+1     ;TYPE OF END ELEMENT
Q504=+0.5   ;SIZE OF END ELEMENT
Q463=+3     ;MAX. CUTTING DEPTH
Q478=+0.3   ;ROUGHING FEED RATE
Q483=+0.4   ;OVERSIZE FOR DIAMETER
Q484=+0.2   ;OVERSIZE IN Z
Q505=+0.2   ;FINISHING FEED RATE
Q506=+0     ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

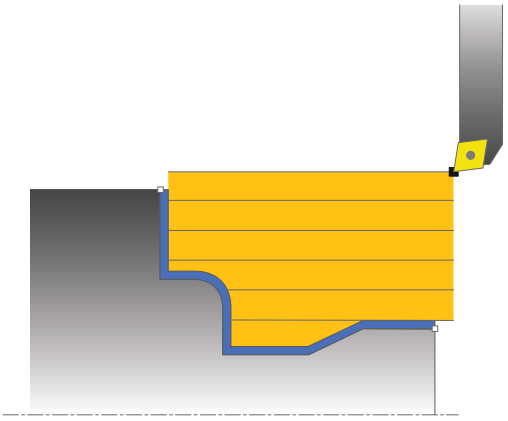
14.11 TURN CONTOUR LONGITUDINAL (Cycle 810, DIN/ISO: G812)

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 coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool 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 control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over 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 was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The tool position at cycle call (cycle start point) influences the area to be machined.

The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

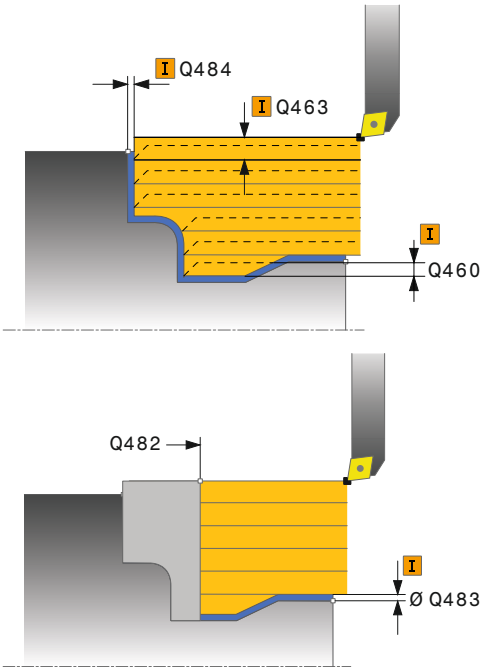
Also refer to the fundamentals of turning cycles (see Page 453).

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

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters

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



- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q487 Allow plunging (0/1)?**: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

```
9 CYCL DEF 14.0 CONTOUR GEOMETRY
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 810 TURN CONTOUR LONG.
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q499=+0     ;REVERSE CONTOUR
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q487=+1     ;PLUNGE
   Q488=+0     ;PLUNGING FEED RATE
   Q479=+0     ;CONTOUR MACHINING LIMIT
   Q480=+0     ;DIAMETER LIMIT VALUE
   Q482=+0     ;LIMIT VALUE Z
   Q506=+0     ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-35
20 RND R5
21 L X+50 Z-40
22 L Z-55
23 CC X+60 Z-55
24 C X+60 Z-60
25 L X+100
26 LBL 0
```

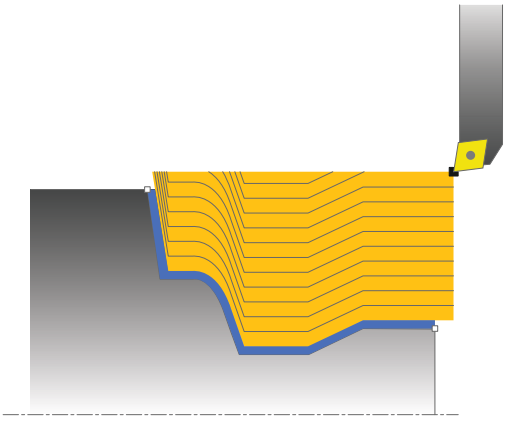
14.12 CONTOUR-PARALLEL TURNING (Cycle 815, DIN/ISO: G815)

Application

This cycle enables you to run 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 with roughing is contour-parallel.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and end point. The cut is performed in contour-parallel mode at the defined feed rate **Q478**.
3 The control returns the tool at the defined feed rate back to the starting position in the X coordinate.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool 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 control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The tool position at cycle call (cycle start point) influences the area to be machined.

The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

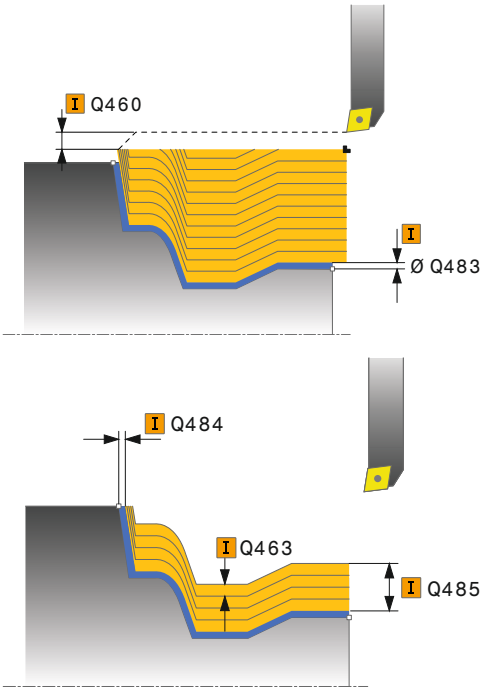
Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

Also refer to the fundamentals of turning cycles (see Page 453).

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

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q485 Allowance for workpiece blank?** (incremental): Contour-parallel oversize for the defined contour
- **Q486 Type of cut lines (=0/1)?**: Define the type of cutting lines:
  0: Cuts with constant chip cross section
  1: Equidistant distribution of the cuts
- **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
  2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute



Example

```
11 CYCL DEF 815  CONTOUR-PAR.
   TURNING
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q485=+5     ;ALLOWANCE ON BLANK
   Q486=+0     ;INTERSECTING LINES
   Q499=+0     ;REVERSE CONTOUR
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=0.3    ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

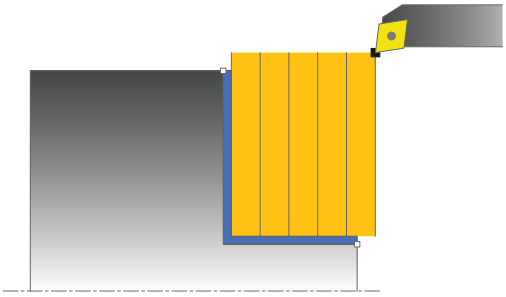
14.13 TURN SHOULDER FACE (Cycle 821, DIN/ISO: G821)

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 machines the area from the cycle starting point to the end point defined in the cycle.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 The control performs a paraxial infeed movement at rapid traverse.
3 The control finishes the contour of the finished part at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

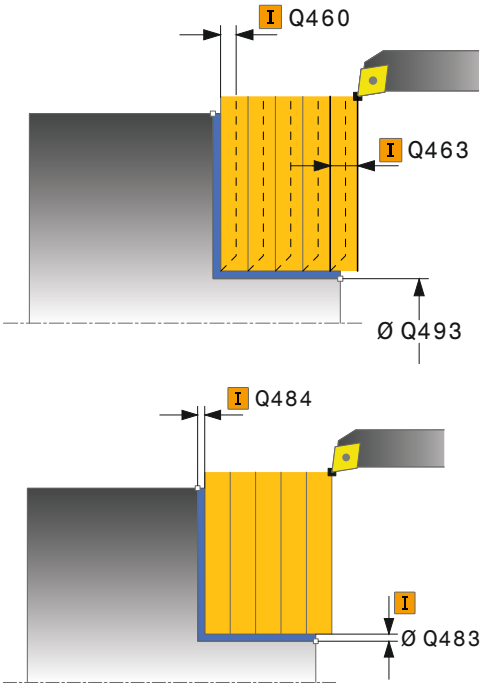
The tool position at cycle call (cycle start point) influences the area to be machined.

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

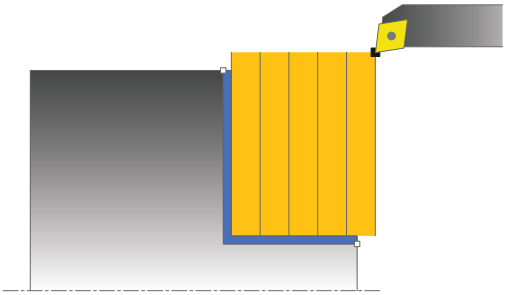


Example

```
11 CYCL DEF 821  SHOULDER, FACE
    Q215=+0      ;MACHINING OPERATION
    Q460=+2      ;SAFETY CLEARANCE
    Q493=+30     ;DIAMETER AT CONTOUR
                  END
    Q494=-5      ;CONTOUR END IN Z
    Q463=+3      ;MAX. CUTTING DEPTH
    Q478=+0.3    ;ROUGHING FEED RATE
    Q483=+0.4    ;OVERSIZE FOR DIAMETER
    Q484=+0.2    ;OVERSIZE IN Z
    Q505=+0.2    ;FINISHING FEED RATE
    Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.14 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 control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control performs a paraxial infeed movement at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

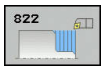
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle start point) influences the area to be machined.

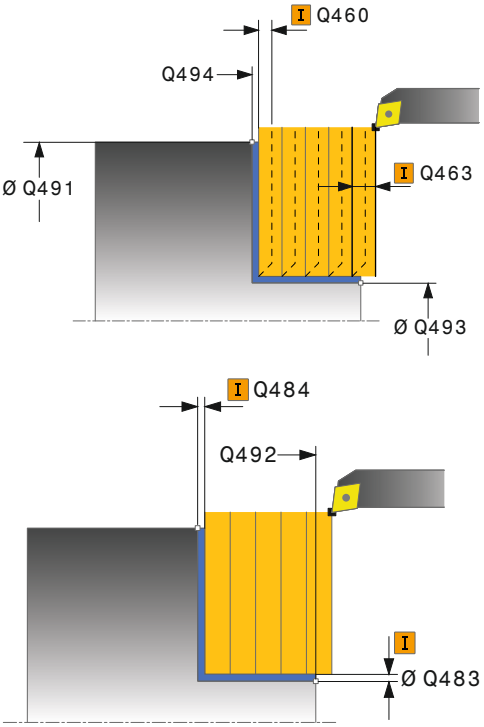
If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of the face?**: Angle between the level surface and the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius



- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of circumferen. surface?**: Angle between the circumferential surface and the rotary axis
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

```
11 CYCL DEF 822  SHOULDER, FACE. EXT.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR START
   Q492=+0      ;CONTOUR START IN Z
   Q493=+30     ;DIAMETER AT CONTOUR END
   Q494=-15     ;CONTOUR END IN Z
   Q495=+0      ;ANGLE OF FACE
   Q501=+1      ;TYPE OF STARTING ELEMENT
   Q502=+0.5    ;SIZE OF STARTING ELEMENT
   Q500=+1.5    ;RADIUS OF CONTOUR EDGE
   Q496=+5      ;ANGLE OF CIRCUM. SURFACE
   Q503=+1      ;TYPE OF END ELEMENT
   Q504=+0.5    ;SIZE OF END ELEMENT
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

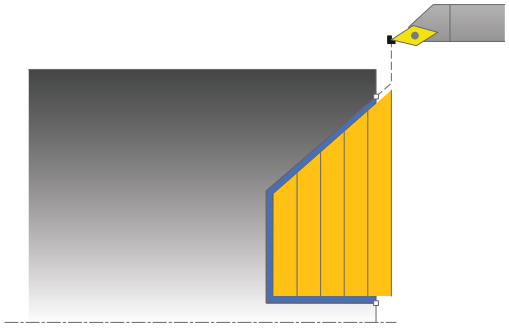
14.15 TURN TRAVERSE PLUNGE (Cycle 823, DIN/ISO: G823)

Application

This cycle enables you to run face turning of plunging 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 control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

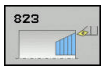
The tool position at cycle call (cycle start point) influences the area to be machined.

The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

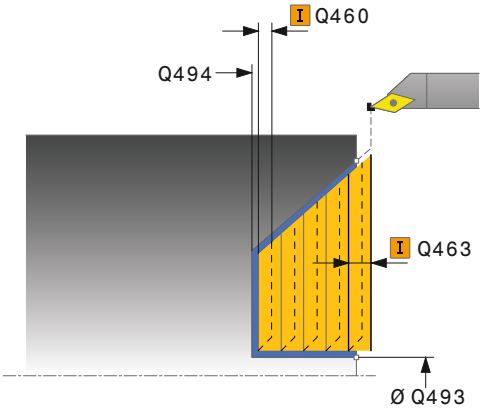
If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

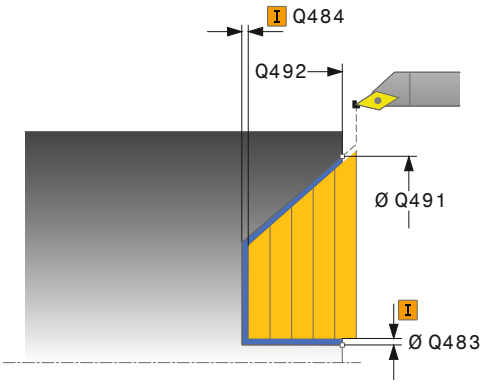
Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°



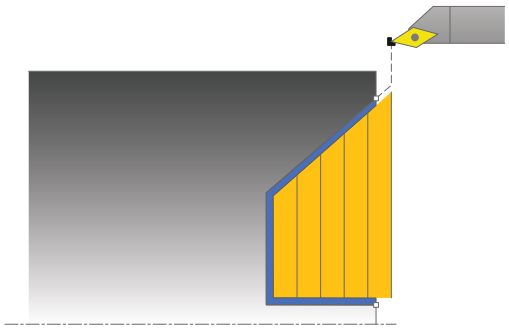


Example

```
11 CYCL DEF 823  TURN TRANSVERSE PLUNGE
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR START
   Q492=+0      ;CONTOUR START IN Z
   Q493=+20     ;DIAMETER AT CONTOUR END
   Q494=-5      ;CONTOUR END IN Z
   Q495=+60     ;ANGLE OF SIDE
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

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

Application



This cycle enables you to run face turning of plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The tool position at cycle call (cycle start point) influences the area to be machined.

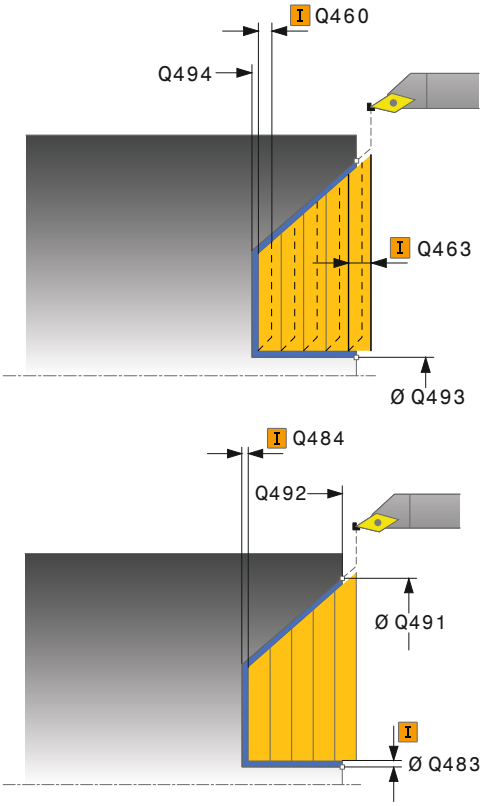
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q491 Diameter at contour start?**: X coordinate of the starting point for the plunging path (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius



- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of circumferen. surface?**: Angle between the circumferential surface and the rotary axis
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

```
11 CYCL DEF 824  TURN PLUNGE TRANSVERSE EXT.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR START
   Q492=+0      ;CONTOUR START IN Z
   Q493=+20     ;DIAMETER AT CONTOUR END
   Q494=-10     ;CONTOUR END IN Z
   Q495=+70     ;ANGLE OF SIDE
   Q501=+1      ;TYPE OF STARTING ELEMENT
   Q502=+0.5    ;SIZE OF STARTING ELEMENT
   Q500=+1.5    ;RADIUS OF CONTOUR EDGE
   Q496=+0      ;ANGLE OF FACE
   Q503=+1      ;TYPE OF END ELEMENT
   Q504=+0.5    ;SIZE OF END ELEMENT
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q506=+0      ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

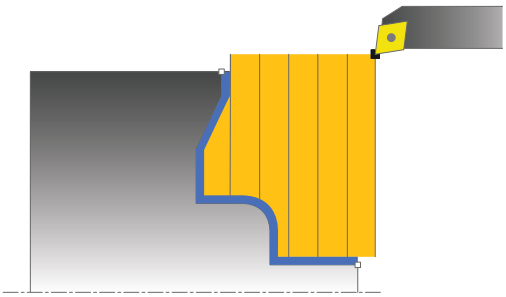
14.17 TURN CONTOUR TRANSVERSE (Cycle 820, DIN/ISO: G820)

Application

This cycle enables you to run face 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 coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction. The transverse cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool 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 control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over 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 was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The tool position at cycle call (cycle start point) influences the area to be machined.

The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

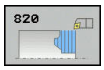
Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

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

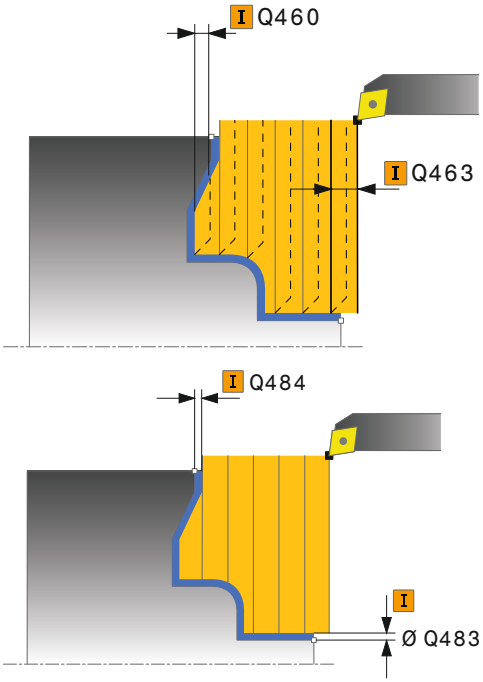
If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Also refer to the fundamentals of turning cycles (see Page 453).

Cycle parameters



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



- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q487 Allow plunging (0/1)?**: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

```
9 CYCL DEF 14.0 CONTOUR GEOMETRY
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 820 TURN CONTOUR TRANSV.
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q499=+0     ;REVERSE CONTOUR
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q487=+1     ;PLUNGE
   Q488=+0     ;PLUNGING FEED RATE
   Q479=+0     ;CONTOUR MACHINING LIMIT
   Q480=+0     ;DIAMETER LIMIT VALUE
   Q482=+0     ;LIMIT VALUE Z
   Q506=+0     ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+75 Z-20
17 L X+50
18 RND R2
19 L X+20 Z-25
20 RND R2
21 L Z+0
22 LBL 0
```

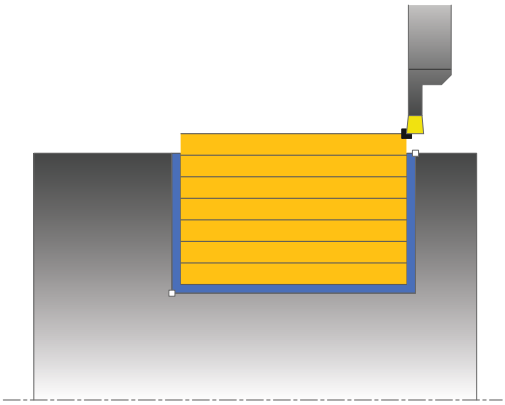
14.18 SIMPLE REC. TURNG., RADIAL DIR. (Cycle 841, DIN/ISO: G841)

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 control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines 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 control lifts off the tool to the set-up clearance, retracts it 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 control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

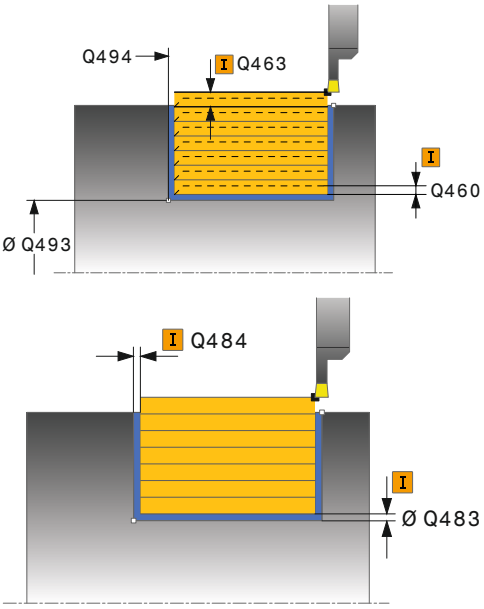
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

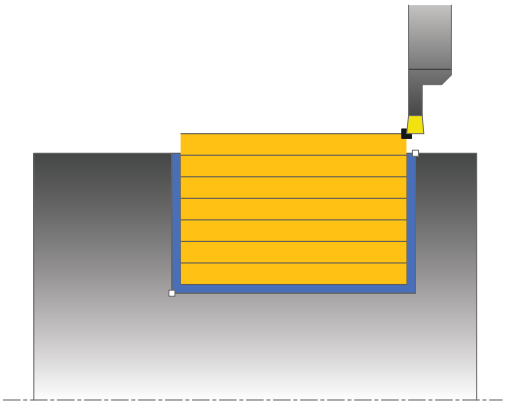


Example

```
11 CYCL DEF 841  SIMPLE REC. TURNG.,
   RADIAL DIR.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q493=+50     ;DIAMETER AT CONTOUR
                 END
   Q494=-50     ;CONTOUR END IN Z
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q463=+2      ;MAX. CUTTING DEPTH
   Q507=+0      ;MACHINING DIRECTION
   Q508=+0      ;OFFSET WIDTH
   Q509=+0      ;DEPTH COMPENSATION
   Q488=+0      ;PLUNGING FEED RATE
12 L  X+75 Y+0  Z-25 FMAX  M303
13 CYCL CALL
```

14.19 EXPANDED RECESS TURNING, RADIAL (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 control uses the position of the tool at cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 Diameter at contour start**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines 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 control lifts off the tool to the set-up clearance, retracts it 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 control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 DIAMETER AT CONTOUR START**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

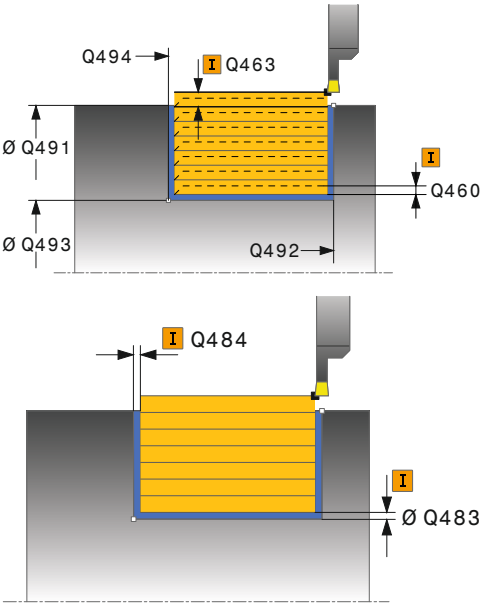
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the perpendicular to the rotary axis.
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the perpendicular to the rotary axis.
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)



Example

11 CYCL DEF 842 EXPND. RECESS, RADL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF SECOND SIDE

- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

```
Q503=+1     ;TYPE OF END ELEMENT
Q504=+0.5   ;SIZE OF END ELEMENT
Q478=+0.3   ;ROUGHING FEED RATE
Q483=+0.4   ;OVERSIZE FOR DIAMETER
Q484=+0.2   ;OVERSIZE IN Z
Q505=+0.2   ;FINISHING FEED RATE
Q463=+2     ;MAX. CUTTING DEPTH
Q507=+0     ;MACHINING DIRECTION
Q508=+0     ;OFFSET WIDTH
Q509=+0     ;DEPTH COMPENSATION
Q488=+0     ;PLUNGING FEED RATE
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.20 SIMPLE RECESS TURNING, AXIAL (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 control uses the tool position as cycle starting point when the cycle is called. The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines the area between the starting position and the end point in transverse 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 control lifts off the tool to the set-up clearance, retracts it 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 control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

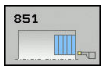
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

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

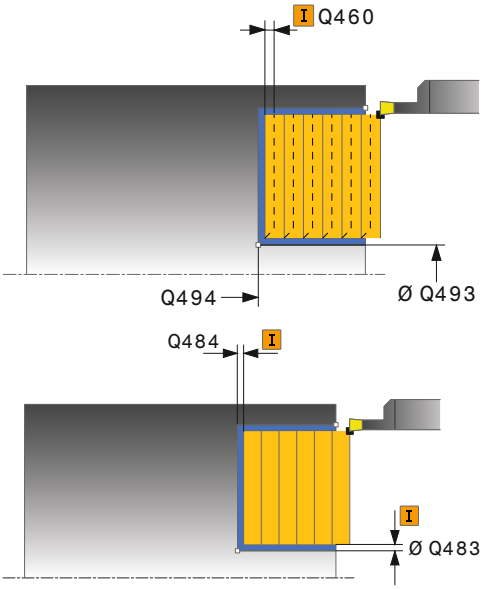
From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

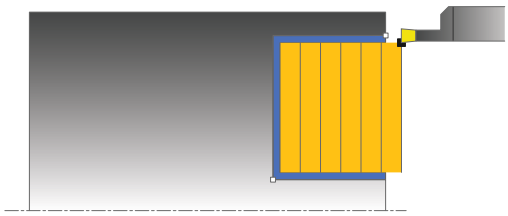


Example

```
11 CYCL DEF 851  SIMPLE REC TURNG, AX
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q493=+50     ;DIAMETER AT CONTOUR END
   Q494=-10     ;CONTOUR END IN Z
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q463=+2      ;MAX. CUTTING DEPTH
   Q507=+0      ;MACHINING DIRECTION
   Q508=+0      ;OFFSET WIDTH
   Q509=+0      ;DEPTH COMPENSATION
   Q488=+0      ;PLUNGING FEED RATE
12 L  X+65 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.21 ENHANCED RECESS TURNING, AXIAL (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 are alternatively performed. The machining process thus requires a minimum of retraction and infeed movements. Extended 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 control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in transverse 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 control lifts off the tool to the set-up clearance, retracts it 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 control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

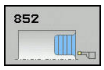
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

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

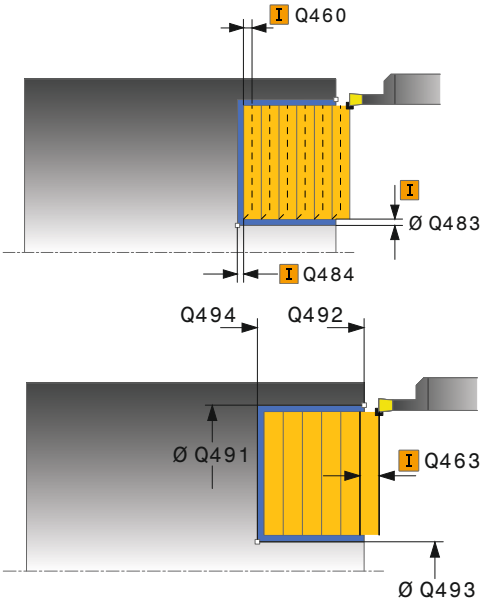
From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the parallel line to the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the parallel line to the rotary axis
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)



Example

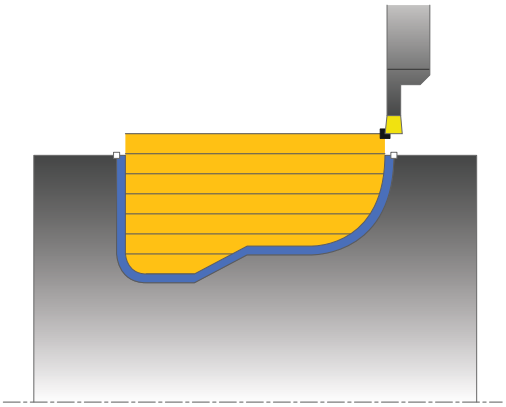
11 CYCL DEF 852 ENH.REC.TURNING, AX.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF SECOND SIDE

- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

```
Q503=+1     ;TYPE OF END ELEMENT
Q504=+0.5   ;SIZE OF END ELEMENT
Q478=+0.3   ;ROUGHING FEED RATE
Q483=+0.4   ;OVERSIZE FOR DIAMETER
Q484=+0.2   ;OVERSIZE IN Z
Q505=+0.2   ;FINISHING FEED RATE
Q463=+2     ;MAX. CUTTING DEPTH
Q507=+0     ;MACHINING DIRECTION
Q508=+0     ;OFFSET WIDTH
Q509=+0     ;DEPTH COMPENSATION
Q488=+0     ;PLUNGING FEED RATE
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.22 CONTOUR RECESS TURNING, RADIAL (Cycle 840, DIN/ISO: G840)

Application



This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

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 coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the X coordinate of the starting point is less than the contour starting point, the control positions the tool in the X coordinate to the contour starting point and begins the cycle there.

1 The control positions the tool at rapid traverse in the Z coordinate (first recessing position).
2 The control performs a recessing traverse until the first plunging depth is reached.
3 The control machines 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 control lifts off the tool to the set-up clearance, retracts it 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 control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over 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 was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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 the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

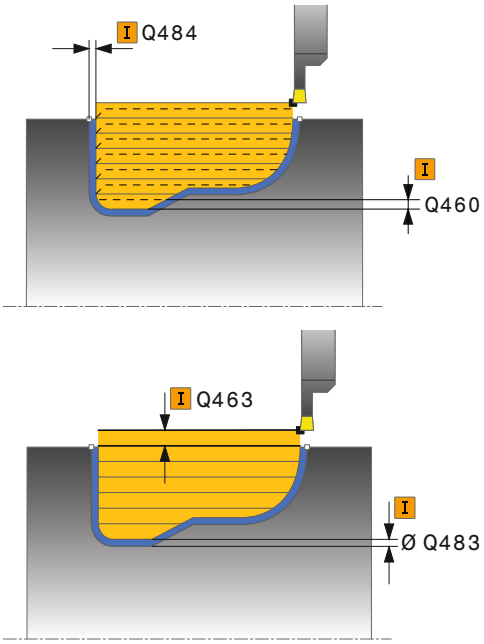
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction



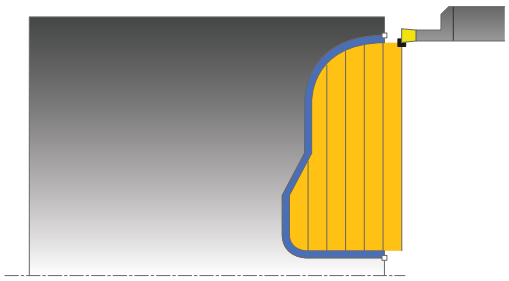
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q499 Reverse contour (0=no/1=yes)?**: Machining direction:
  0: Contour machined in the programmed direction
  1: in reverse direction to the programmed direction

Example

```
9 CYCL DEF 14.0 CONTOUR
10  CYCL DEF 14.1  CONTOUR LABEL2
11 CYCL DEF 840  RECESS TURNG, RADIAL
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q478=+0.3   ;ROUGHING FEED RATE
    Q488=+0     ;PLUNGING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q479=+0     ;CONTOUR MACHINING LIMIT
    Q480=+0     ;DIAMETER LIMIT VALUE
    Q482=+0     ;LIMIT VALUE Z
    Q463=+2     ;MAX. CUTTING DEPTH
    Q507=+0     ;MACHINING DIRECTION
    Q508=+0     ;OFFSET WIDTH
    Q509=+0     ;DEPTH COMPENSATION
    Q499=+0     ;REVERSE CONTOUR
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z-10
17 L  X+40 Z-15
18 RND  R3
19 CR  X+40 Z-35  R+30 DR+
18 RND  R3
20 L  X+60 Z-40
21 LBL  0
```

14.23 CONTOUR RECESS TURNING, AXIAL (Cycle 850, DIN/ISO: G850)

Application



This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

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 coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control positions the tool at rapid traverse in the X coordinate (first recessing position).
2 The control performs a recessing traverse until the first plunging depth is reached.
3 The control machines the area between the starting position and the end point in transverse 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 control lifts off the tool to the set-up clearance, retracts it 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 control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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 the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

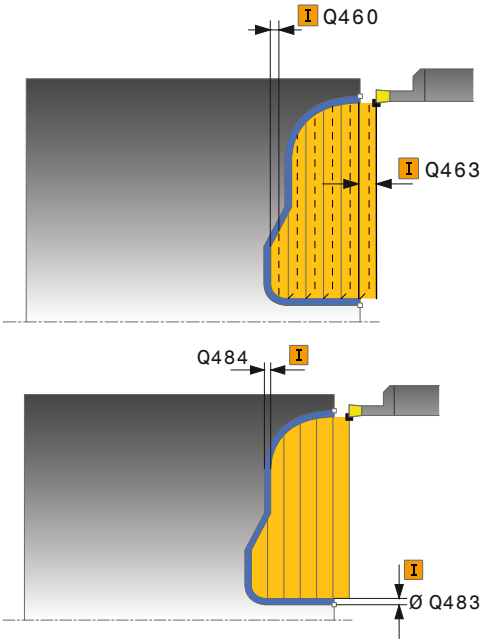
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction



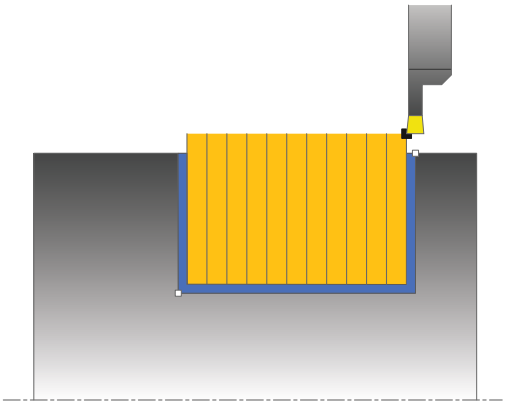
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range: 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q499 Reverse contour (0=no/1=yes)?**: Machining direction:
  0: Contour machined in the programmed direction
  1: in reverse direction to the programmed direction

Example

```
9 CYCL DEF 14.0 CONTOUR GEOMETRY
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 850 RECESS TURNG, AXIAL
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q478=+0.3   ;ROUGHING FEED RATE
    Q488=0      ;PLUNGING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q479=+0     ;CONTOUR MACHINING LIMIT
    Q480=+0     ;DIAMETER LIMIT VALUE
    Q482=+0     ;LIMIT VALUE Z
    Q463=+2     ;MAX. CUTTING DEPTH
    Q507=+0     ;MACHINING DIRECTION
    Q508=+0     ;OFFSET WIDTH
    Q509=+0     ;DEPTH COMPENSATION
    Q499=+0     ;REVERSE CONTOUR
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-15
20 L Z+0
21 LBL 0
```

14.24 SIMPLE RECESSING, RADIAL (Cycle 861, DIN/ISO: G861)

Application

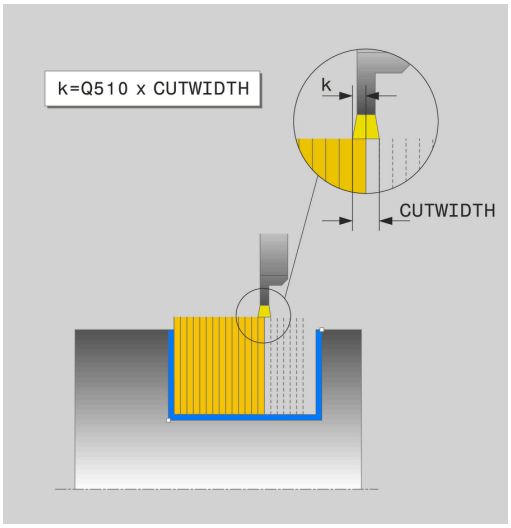


This cycle enables you to radially cut in right-angled slots.

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

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

Finishing cycle run

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes half the slot width at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes half the slot width at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

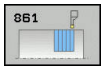
This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

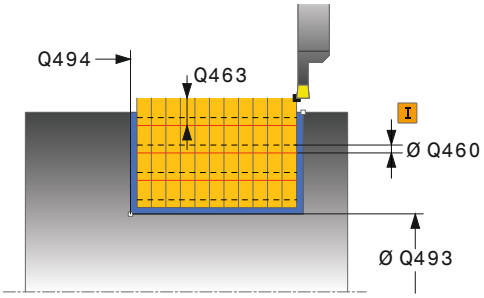
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

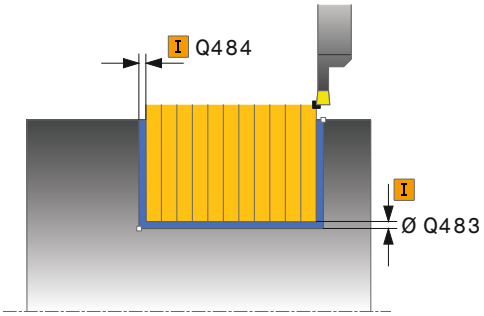
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor **Q510** influences the stepover of the tool when roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0.001 to 1





Example

```
11 CYCL DEF 861  SIMPLE RECESS,
   RADL.
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q493=+50    ;DIAMETER AT CONTOUR
               END
   Q494=-50    ;CONTOUR END IN Z
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
```

- **Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate **Q478** so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0.001 to 150
- **Q462 Retraction behavior (0/1)? Q462** defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q463=+0      ;LIMIT TO DEPTH
    Q510=+0.8    ;RECESSING OVERLAP
    Q511=+100    ;FEED RATE FACTOR
    Q462=0       ;RETRACTION MODE
    Q211=3       ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z-25 FMAX  M303
13 CYCL CALL
```

14.25 EXPANDED RECESSING, RADIAL (Cycle 862, DIN/ISO: G862)

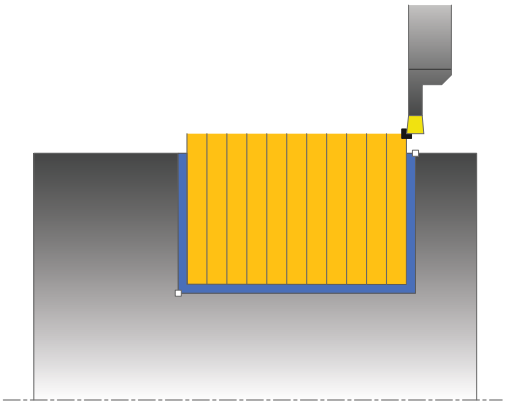
Application

This cycle enables you to radially cut in slots. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

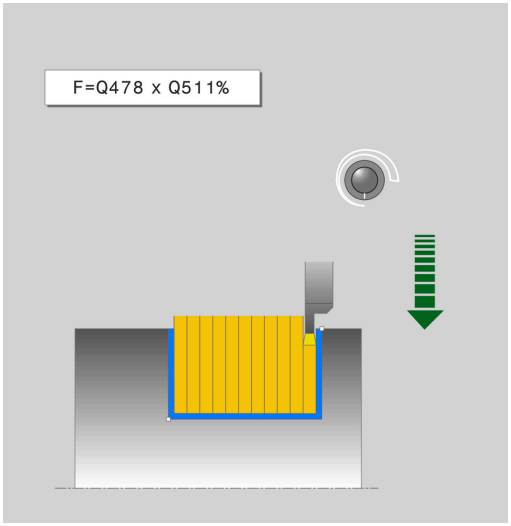
You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.



Roughing cycle run

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



Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

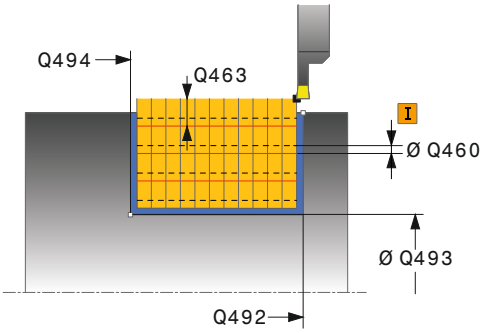
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

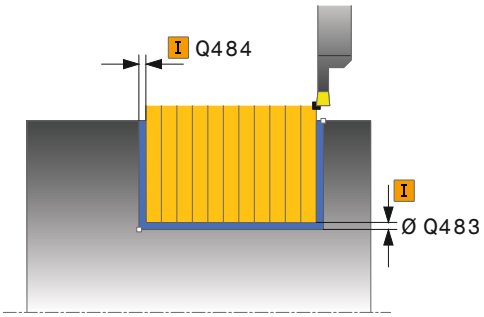
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the perpendicular to the rotary axis.
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the perpendicular to the rotary axis.
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)





Example

11 CYCL DEF 862 EXPND. RECESS, RADL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE

- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor **Q510** influences the stepover of the tool when roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0.001 to 1
- **Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate **Q478** so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0.001 to 150
- **Q462 Retraction behavior (0/1)? Q462** defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

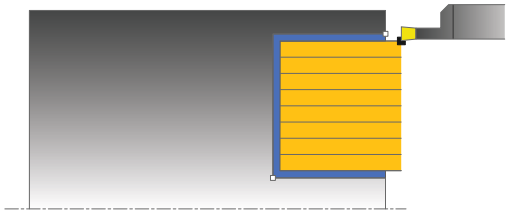
```
Q496=+5     ;ANGLE OF SECOND SIDE
Q503=+1     ;TYPE OF END ELEMENT
Q504=+0.5   ;SIZE OF END ELEMENT
Q478=+0.3   ;ROUGHING FEED RATE
Q483=+0.4   ;OVERSIZE FOR DIAMETER
Q484=+0.2   ;OVERSIZE IN Z
Q505=+0.2   ;FINISHING FEED RATE
Q463=+0     ;LIMIT TO DEPTH
Q510=0.8    ;RECESSING OVERLAP
Q511=+100   ;FEED RATE FACTOR
Q462=+0     ;RETRACTION MODE
Q211=3      ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.26 SIMPLE RECESSING, AXIAL (Cycle 871, DIN/ISO: G871)

Application

This cycle enables you to perform axial recessing of 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 control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

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

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

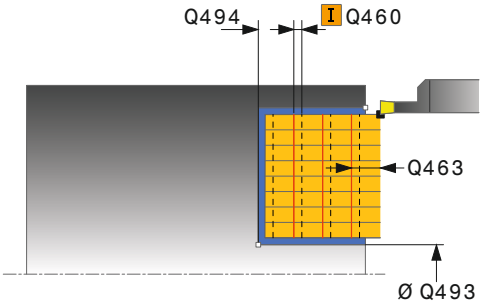
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

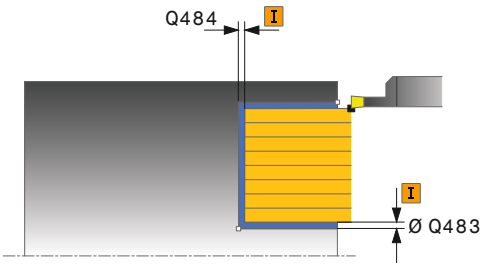
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor **Q510** influences the stepover of the tool when roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0.001 to 1





Example

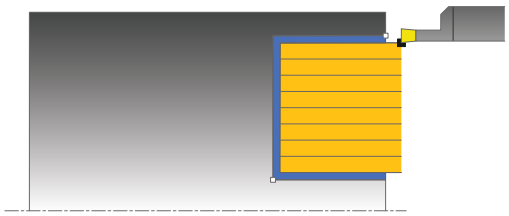
11 CYCL DEF 871 SIMPLE RECESS, AXIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-10	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+0	;LIMIT TO DEPTH

- **Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate **Q478** so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0.001 to 150
- **Q462 Retraction behavior (0/1)? Q462** defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q510=+0.8  ;RECESSING OVERLAP
    Q511=+100  ;FEED RATE FACTOR
    Q462=0     ;RETRACTION MODE
    Q211=3     ;DWELL TIME IN REVS
12 L  X+65 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.27 EXPANDED RECESSING, AXIAL (Cycle 872, DIN/ISO: G872)

Application



This cycle enables you to perform axial recessing of slots (face recessing). Extended 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 control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

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

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control retracts the tool at rapid traverse.
4 The control positions the tool at rapid traverse to the second slot side.
5 The control finishes the side wall of the slot at the defined feed rate **Q505**.
6 The control finishes one half of the slot at the defined feed rate.
7 The control positions the tool at rapid traverse to the first side.
8 The control finishes the other half of the slot at the defined feed rate.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

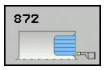
This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

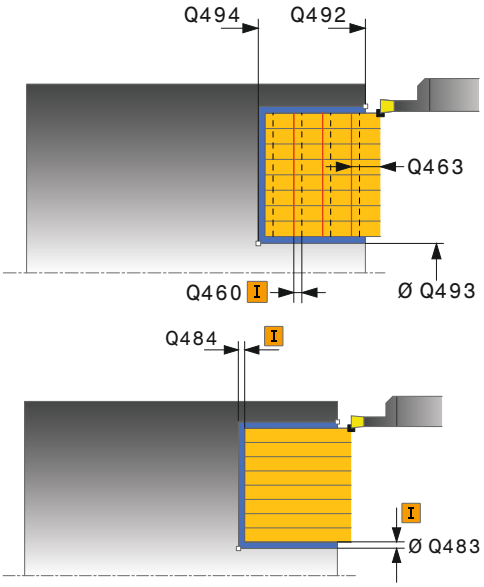
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the parallel line to the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that of the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the parallel line to the rotary axis



Example

```
11 CYCL DEF 871  EXPND. RECESS, AXIAL
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q491=+75     ;DIAMETER AT CONTOUR START
   Q492=-20     ;CONTOUR START IN Z
   Q493=+50     ;DIAMETER AT CONTOUR END
   Q494=-50     ;CONTOUR END IN Z
   Q495=+5      ;ANGLE OF SIDE
```

- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor **Q510** influences the stepover of the tool when roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0.001 to 1

```
Q501=+1      ;TYPE OF STARTING ELEMENT
Q502=+0.5    ;SIZE OF STARTING ELEMENT
Q500=+1.5    ;RADIUS OF CONTOUR EDGE
Q496=+5      ;ANGLE OF SECOND SIDE
Q503=+1      ;TYPE OF END ELEMENT
Q504=+0.5    ;SIZE OF END ELEMENT
Q478=+0.3    ;ROUGHING FEED RATE
Q483=+0.4    ;OVERSIZE FOR DIAMETER
Q484=+0.2    ;OVERSIZE IN Z
Q505=+0.2    ;FINISHING FEED RATE
Q463=+0      ;LIMIT TO DEPTH
Q510=+0.08;RECESSING OVERLAP
Q511=+100    ;FEED RATE FACTOR
Q462=0       ;RETRACTION MODE
Q211=3       ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

- **Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate **Q478** so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0.001 to 150
- **Q462 Retraction behavior (0/1)? Q462** defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

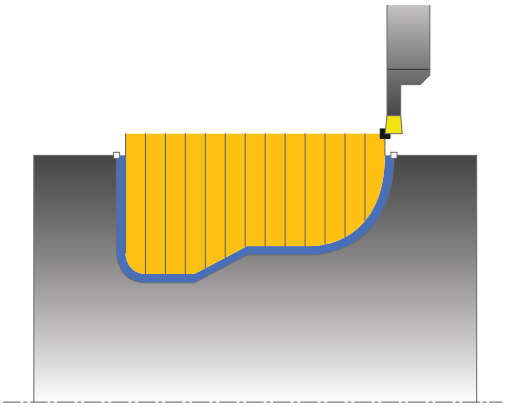
14.28 CONTOUR RECESSING, RADIAL (Cycle 860, DIN/ISO: G860)

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 coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

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

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over 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 was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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)

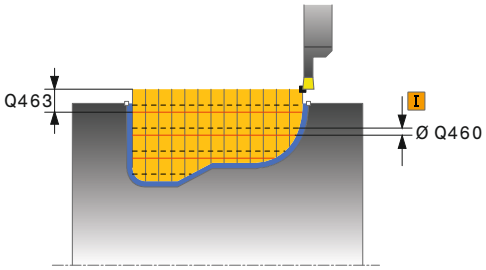
Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

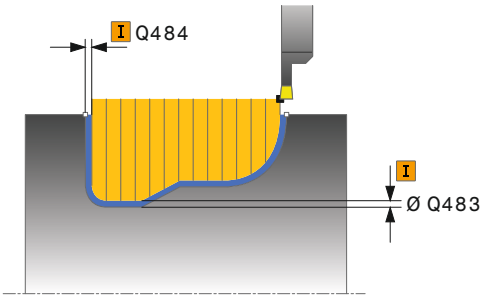
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut





Example

```
9  CYCL DEF 14.0  CONTOUR GEOMETRY
10  CYCL DEF 14.1  CONTOUR LABEL2
11 CYCL DEF 860  CONT. RECESS,
   RADIAL
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q478=+0.3   ;ROUGHING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
```

- **Q510 Overlap factor for recess width?** Factor **Q510** influences the stepover of the tool when roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0.001 to 1
- **Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate **Q478** so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0.001 to 150
- **Q462 Retraction behavior (0/1)? Q462** defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

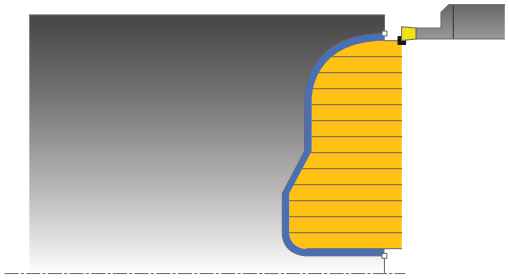
```
Q479=+0     ;CONTOUR MACHINING LIMIT
Q480=+0     ;DIAMETER LIMIT VALUE
Q482=+0     ;LIMIT VALUE Z
Q463=+0     ;LIMIT TO DEPTH
Q510=0.08   ;RECESSING OVERLAP
Q511=+100   ;FEED RATE FACTOR
Q462=+0     ;RETRACTION MODE
Q211=3      ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z-20
17 L  X+45
18 RND  R2
19 L  X+40 Z-25
20 L  Z+0
21 LBL  0
```

14.29 CONTOUR RECESSING, AXIAL (Cycle 870, DIN/ISO: G870)

Application

This cycle enables you to perform axial recessing of 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 control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

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

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over 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 was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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)

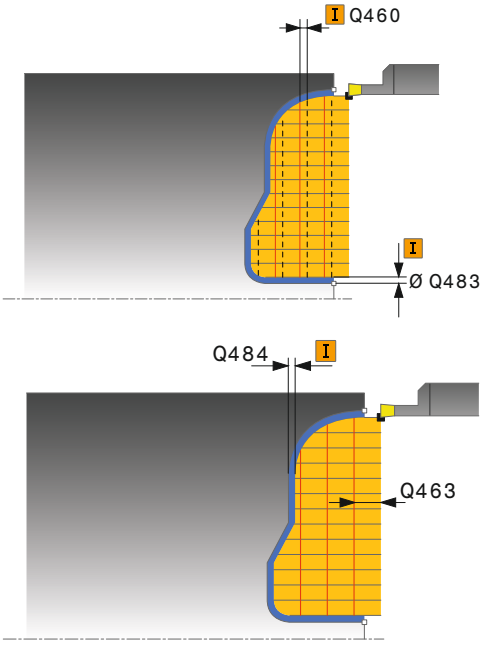
Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

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

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor **Q510** influences the stepover of the tool when roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0.001 to 1



Example

```
9 CYCL DEF 14.0 CONTOUR GEOMETRY
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 870 CONT. RECESS, AXIAL
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q478=+0.3   ;ROUGHING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q479=+0     ;CONTOUR MACHINING LIMIT
    Q480=+0     ;DIAMETER LIMIT VALUE
    Q482=+0     ;LIMIT VALUE Z
```

- **Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate **Q478** so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0.001 to 150
- **Q462 Retraction behavior (0/1)? Q462** defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q463=+0     ;LIMIT TO DEPTH
    Q510=0.8    ;RECESSING OVERLAP
    Q511=+100 ;FEED RATE FACTOR
    Q462=+0     ;RETRACTION MODE
    Q211=3      ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z+0
17 L  Z-10
18 RND  R5
19 L  X+40 Z-15
20 L  Z+0
21 LBL  0
```

14.30 THREAD, LONGITUDINAL (Cycle 831, DIN/ISO: G831)

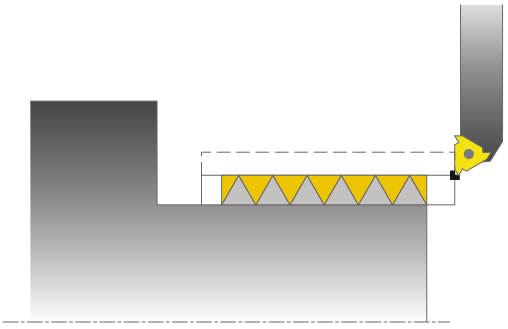
Application

This cycle enables you to run longitudinal turning of threads.

You can machine single threads or multi-threads with this 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 control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a paraxial longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g. to machine internal and external threads with only one tool. If with such a machine you wish to use an outside machine tool for inside machining, you can execute machining in the negative X- diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Danger of collision!

If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

- Program the **Q467** angle of infeed to be less than the side angle of the thread.

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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

The number of threads for thread cutting is limited to 500.

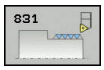
The control uses the set-up clearance **Q460** as approach length. The approach path must be long enough for the feed axes to be accelerated to the required velocity.

The control uses the thread pitch as idle travel path. The idle travel distance must be long enough to decelerate the feed axes.

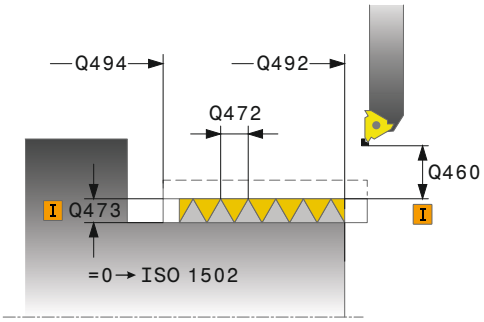
Parameters are available for approach and overrun in Cycle 832 THREAD EXTENDED.

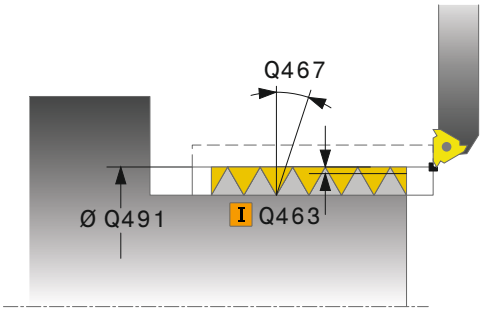
When the control cuts a thread, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

Cycle parameters



- **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Q460 Setup clearance?**: Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) until the synchronized feed rate is reached.
- **Q491 Thread diameter?**: Define the nominal diameter of the thread.
- **Q472 Thread pitch?**: Pitch of the thread
- **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Q492 Contour start in Z?**: Z coordinate of the starting point
- **Q494 Contour end in Z?**: Z coordinate of the end point including the runout of the thread **Q474**.
- **Q474 Length of thread runout?** (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**.
- **Q463 Maximum cutting depth?**: Maximum plunging depth in radial direction relative to the radius.
- **Q467 Feed angle?**: Angle for the infeed **Q463**. The reference angle is formed by the perpendicular to the rotary axis.
- **Q468 Infeed type (0/1)?**: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Q470 Starting angle?**: Angle of the turning spindle at which the thread should start.
- **Q475 Number of thread grooves?**: Number of thread grooves
- **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth



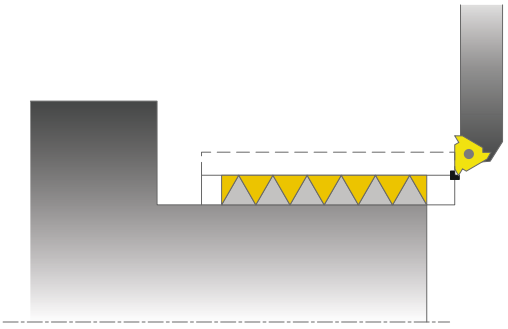


Example

```
11 CYCL DEF 831  THREAD
   LONGITUDINAL
   Q471=+0     ;THREAD POSITION
   Q460=+5     ;SAFETY CLEARANCE
   Q491=+75    ;THREAD DIAMETER
   Q472=+2     ;THREAD PITCH
   Q473=+0     ;DEPTH OF THREAD
   Q492=+0     ;CONTOUR START IN Z
   Q494=-15    ;CONTOUR END IN Z
   Q474=+0     ;THREAD RUN-OUT
   Q463=+0.5   ;MAX. CUTTING DEPTH
   Q467=+30    ;ANGLE OF INFEED
   Q468=+0     ;TYPE OF INFEED
   Q470=+0     ;STARTING ANGLE
   Q475=+30    ;NUMBER OF STARTS
   Q476=+30    ;NUMBER OF AIR CUTS
12 L  X+80 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.31 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 a longitudinal thread or transversal 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 the approach length and the idle travel distance define a path in which feed axes can be accelerated and 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 control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g. to machine internal and external threads with only one tool. If with such a machine you wish to use an outside machine tool for inside machining, you can execute machining in the negative X- diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Danger of collision!

If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

- Program the **Q467** angle of infeed to be less than the side angle of the thread.

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

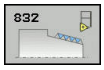
Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.

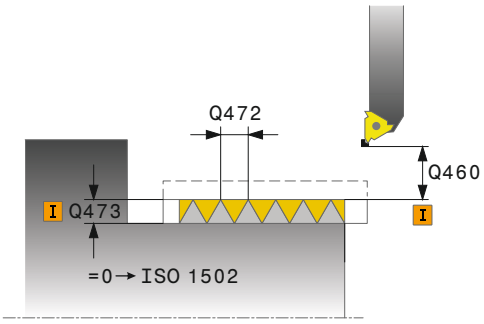
The overrun path (**Q466**) must be long enough to decelerate the feed axes.

When the control cuts a thread, 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).

Cycle parameters



- **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Q461 Thread orientation (0/1)?**: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)
- **Q460 Set-up clearance?**: Set-up clearance perpendicular to the thread pitch.
- **Q472 Thread pitch?**: Pitch of the thread
- **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Q464 Dimens. type taper (0-4)?**: Define the type of dimension for the taper contour:
  0: Via starting point and end point
  1: Via end point, start-X and taper angle
  2: Via end point, start-Z and taper angle
  3: Via starting point, end-X and taper angle
  4: Via starting point, end-Z and taper angle
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point
- **Q493 Diameter at end of contour?**: X coordinate of the end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the end point
- **Q469 Taper angle (diameter)?** Taper angle of the contour
- **Q474 Length of thread runout?** (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**.



Example

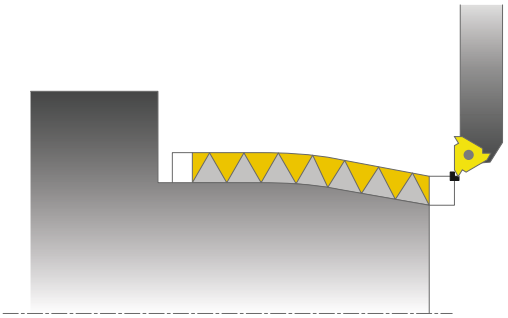
11 CYCL DEF 832 THREAD EXTENDED	
Q471=+0	;THREAD POSITION
Q461=+0	;THREAD ORIENTATION
Q460=+2	;SAFETY CLEARANCE
Q472=+2	;THREAD PITCH
Q473=+0	;DEPTH OF THREAD
Q464=+0	;DIMENSION TYPE TAPER
Q491=+100	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+110	;DIAMETER AT CONTOUR END
Q494=-35	;CONTOUR END IN Z
Q469=+0	;TAPER ANGLE
Q474=+0	;THREAD RUN-OUT
Q465=+4	;STARTING PATH
Q466=+4	;OVERRUN PATH
Q463=+0.5	;MAX. CUTTING DEPTH
Q467=+30	;ANGLE OF INFEED

- **Q465 Starting path?** (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside the defined thread contour.
- **Q466 Overrun path?**: Length of the path in pitch direction on which the feed axes are decelerated. The idle travel path is within the defined thread contour.
- **Q463 Maximum cutting depth?**: Maximum plunging depth perpendicular to the thread pitch
- **Q467 Feed angle?**: Angle for the infeed **Q463**. The reference angle is formed by a line that is parallel to the thread pitch
- **Q468 Infeed type (0/1)?**: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Q470 Starting angle?**: Angle of the turning spindle at which the thread should start.
- **Q475 Number of thread grooves?**: Number of thread grooves
- **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth

```
    Q468=+0     ;TYPE OF INFEED
    Q470=+0     ;STARTING ANGLE
    Q475=+30    ;NUMBER OF STARTS
    Q476=+30    ;NUMBER OF AIR CUTS
12 L  X+80 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

14.32 THREAD, CONTOUR-PARALLEL (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 machine single threads or multi-threads with this 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.

NOTICE

Danger of collision!

Cycle 830 runs the overrun **Q466** following the programmed contour. Take the available space into account.

- Clamp the workpiece in such a way that there is no danger of collision if the control extends the contour by **Q466**, **Q467**.

Cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control runs a thread cut parallel to the defined thread contour. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g. to machine internal and external threads with only one tool. If with such a machine you wish to use an outside machine tool for inside machining, you can execute machining in the negative X- diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Danger of collision!

If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

- Program the **Q467** angle of infeed to be less than the side angle of the thread.

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

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.

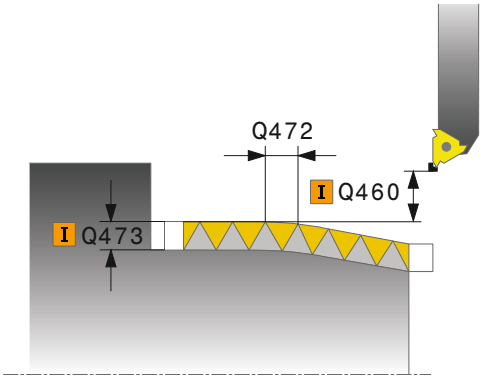
When the control cuts a thread, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

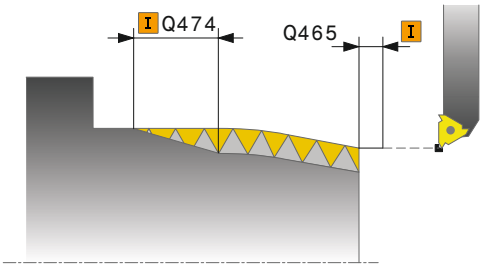
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

- **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Q461 Thread orientation (0/1)?**: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)
- **Q460 Set-up clearance?**: Set-up clearance perpendicular to the thread pitch.
- **Q472 Thread pitch?**: Pitch of the thread
- **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Q474 Length of thread runout?** (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**.



- **Q465 Starting path?** (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside the defined thread contour.
- **Q466 Overrun path?**: Length of the path in pitch direction on which the feed axes are decelerated. The idle travel path is within the defined thread contour.
- **Q463 Maximum cutting depth?**: Maximum plunging depth perpendicular to the thread pitch
- **Q467 Feed angle?**: Angle for the infeed **Q463**. The reference angle is formed by a line that is parallel to the thread pitch
- **Q468 Infeed type (0/1)?**: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Q470 Starting angle?**: Angle of the turning spindle at which the thread should start.
- **Q475 Number of thread grooves?**: Number of thread grooves
- **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth



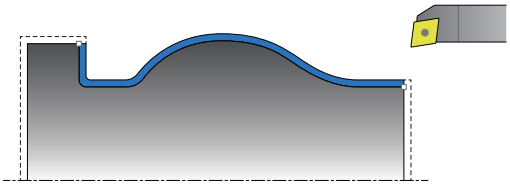
Example

```
9 CYCL DEF 14.0 CONTOUR GEOMETRY
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 830 THREAD CONTOUR-
   PARALLEL
   Q471=+0     ;THREAD POSITION
   Q461=+0     ;THREAD ORIENTATION
   Q460=+2     ;SAFETY CLEARANCE
   Q472=+2     ;THREAD PITCH
   Q473=+0     ;DEPTH OF THREAD
   Q474=+0     ;THREAD RUN-OUT
   Q465=+4     ;STARTING PATH
   Q466=+4     ;OVERRUN PATH
   Q463=+0.5   ;MAX. CUTTING DEPTH
   Q467=+30    ;ANGLE OF INFEED
   Q468=+0     ;TYPE OF INFEED
   Q470=+0     ;STARTING ANGLE
   Q475=+30    ;NUMBER OF STARTS
   Q476=+30    ;NUMBER OF AIR CUTS
12 L  X+80 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z+0
17 L  X+70 Z-30
18 RND  R60
19 L  Z-45
20 LBL  0
```

14.33 TURNING, SIMULTANEOUS FINISHING (Cycle 883, DIN/ISO: G883, (option 158)

Application

Refer to your machine manual!
Option 50 must have been enabled.
Option 158 must have been enabled.



You can use this cycle to machine complex contours that are only accessible with different inclinations. When machining with this cycle, the inclination between tool and workpiece changes. This results in machining operations with at least 3 axes (two linear axes and one rotary axis).

The cycle monitors the workpiece contour with respect to the tool and the tool carrier. The cycle avoids unnecessary tilting movements in order to machine optimum surfaces.
If you want to force tilting movements, you can define inclination angles at the beginning and at the end of the contour. Even if simple contours have to be machined, you can use a large area of the indexable insert to achieve longer tool life.

You define the contour in a subprogram and reference it using Cycle 14 or **SEL CONTOUR**.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The control moves the tool to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. If programmed, the tool traverses to the inclination angle that was calculated by the control based on the minimum and maximum inclination angles you have defined.
3. The control finishes the contour of the finished part (contour starting point to contour end point) simultaneously at the defined feed rate **Q505**.
4. The control retracts the tool at the defined feed rate to the set-up clearance.
5. The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

NOTICE

Danger of collision!

The control does not monitor any collisions between the tool and the workpiece (DCM). Incorrect pre-positioning can also lead to contour damage. Danger of collision during machining!

- Program a suitable pre-position
- Check the machining sequence and the contour with the aid of the graphic simulation as well as by running the program slowly in Program Run, Full Sequence operating mode.

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining.

- Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

Cycle 883 **TURNING SIMULTANEOUS FINISHING** is machine-dependent.

This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

If the tilting axis is not perpendicular to the axis of the turning spindle, an error message will be displayed.

Based on the programmed parameters, the control calculates only **one** collision-free path.

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

Before calling the cycle, you need to program **FUNCTION TCPM** with the **REFPNT TIP-CENTER** tool center point.

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

The inclination angles **Q556** and **Q557** are limited by software limit switches.

If you deactivate monitoring by the software limit switches in the Test Run operating mode, the path might be different from the one that will actually be machined.

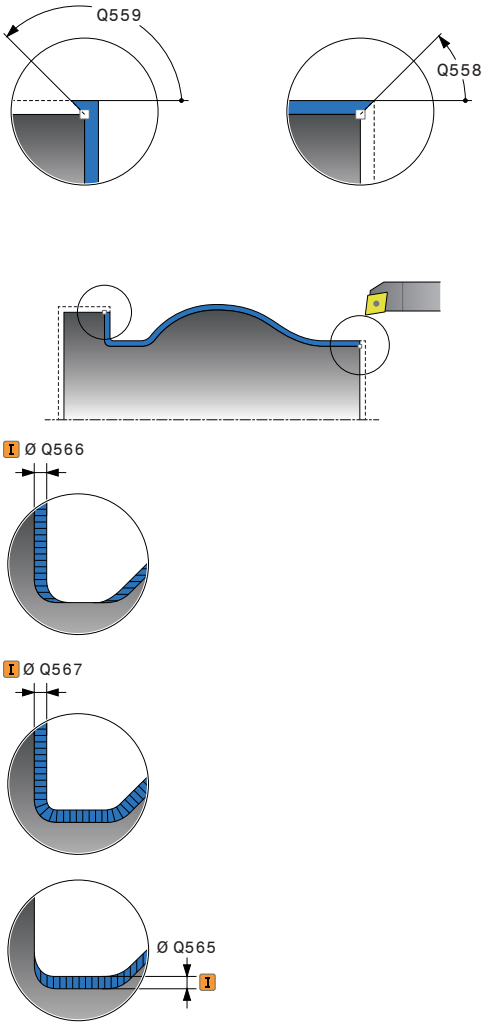
Please note: The smaller the resolution in cycle parameter **Q555** is, the easier will it be to find a solution even in complex situations. The drawback is that the calculation will take more time.

The cycle requires the tool holder geometry. Define it in the KINEMATIC column of the tool table (tool.t). The cycle monitors a 2-D cut with respect to the workpiece contour. The depth of the holder will **not** be monitored

Please note that cycle parameters **Q565** (Finishing allowance in diameter) and **Q566** (Finishing allowance in Z) cannot be combined with **Q567** (Finishing allowance of contour)!

Cycle parameters

- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning.
- **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
  2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- **Q558 Extensn. angle at contour start?**: The contour is extended by a line from the contour starting point with the angle indicated here. The control tries to approach the extension tangentially (WPL-CS)
- **Q559 Extension angle at contour end?**: The contour is extended by a line from the contour end point with the angle indicated here. The control tries to depart from the extension tangentially (WPL-CS)
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute
- **Q556 Minimum angle of inclination?**: Minimum permitted angle of incidence between the tool (Z direction of the tool) and the workpiece (Z direction of the turning spindle)
- **Q557 Maximum angle of inclination?**: Maximum permitted angle of incidence between the tool (Z direction of the tool) and the workpiece (Z direction of the turning spindle)



- **Q555 Stepping angle for calculation?**: Increment for the calculation of possible solutions. Input range: 0.5 to 9.99
- **Q537 Inclin. angle (0=N/1=J/2=S/3=E)?**: Specify whether an inclination angle is active:
  0: No inclination angles active
  1: Inclination angle active
  2: Inclination angle at contour start is active
  3: Inclination angle at the contour end is active
- **Q538 Inclin. angle at contour start?**: Inclination angle at the beginning of the programmed contour (WPL CS)
- **Q539 Inclinatn. angle at contour end?**: Inclination angle at the end of the programmed contour (WPL-CS)
- **Q565 Finishing allowance in diameter** (incremental): Diameter allowance to remain on the contour after finishing.
- **Q566 Finishing allowance in Z?** (incremental): Allowance for the programmed contour in axial direction; material to remain on the contour after finishing.
- **Q567 Finishing allowance of contour?** (incremental): Contour-parallel oversize for the defined contour; material to remain on the contour after finishing.

Example

```
11 CYCL DEF 883  TURNING
   SIMULTANEOUS FINISHING
   Q460=+2     ;SETUP CLEARANCE?
   Q499=+0     ;REVERSE CONTOUR
   Q558=+0     ;EXT:ANGLE CONT.START
   Q559=+90    ;CONTOUR END EXT ANGL
   Q505=+0.2   ;FINISHING FEED RATE
   Q556=-30    ;MIN. INCLINAT. ANGLE
   Q557=+30    ;MAX. INCLINAT. ANGLE
   Q555=+7     ;STEPPING ANGLE
   Q537=+0     ;INCID. ANGLE ACTIVE
   Q538=+0     ;INCLIN. ANGLE START
   Q539=+0     ;INCLINATN. ANGLE END
   Q565=+0     ;FINISHING ALLOW. D.
   Q566=+0     ;FINISHING ALLOW. Z
   Q567=+0     ;FINISH. ALLOW. CONT.
12 L  X+58 Y+0 FMAX  M303
13 L Z+50 FMAX
14 CYCL CALL
```

14.34 Programming example

Example: Gear hobbing

The following NC program illustrates the use of Cycle 880 HOBBING. This programming example illustrates the machining of a helical gear, with Module=2.1.

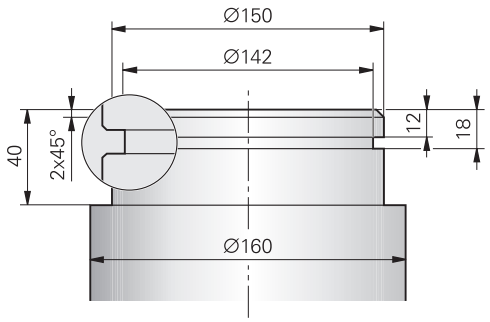
Program run

- Tool call: Gear hob
- Start turning mode
- Approach safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R42 L150	Workpiece blank definition: cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TOOL CALL "GEAR_HOB_D75"	Call the tool
4 FUNCTION MODE TURN	Activate turning mode
5 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	Reset the coordinate system
6 M145	Deactivate M144 if still active
7 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
8 M140 MB MAX	Retract the tool
9 L A+0 R0 FMAX	Set the rotary axis to 0
10 L X+250 Y-250 R0 FMAX	Pre-position the tool in the working plane on the side on which machining will be performed
11 Z+20 R0 FMAX	Pre-position the tool in the spindle axis
12 L M136	Feed rate in mm/rev
13 CYCL DEF 880 GEAR HOBBING	Cycle definition: hobbing
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.	
Q547=+0 ;ANG. OFFSET, SPINDLE	
Q550=+0 ;MACHINING SIDE	
Q533=+0 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	
Q253=+2000 ;F PRE-POSITIONING	
Q260=+20 ;CLEARANCE HEIGHT	
Q553=+10 ;TOOL LENGTH OFFSET	

Q551=+0 ;STARTING POINT IN Z	
Q552=-10 ;END POINT IN Z	
Q463=+1 ;MAX. CUTTING DEPTH	
Q460=2 ;SAFETY CLEARANCE	
Q488=+1 ;PLUNGING FEED RATE	
Q478=+2 ;ROUGHING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	
Q505=+1 ;FINISHING FEED RATE	
14 CYCL CALL M303	Call the cycle, spindle on
15 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	Reset the coordinate system
16 M145	Deactivate the active M144 in the cycle
17 FUNCTION MODE MILL	Activate milling mode
18 M140 MB MAX	Retract the tool in the tool axis
19 L A+0 C+0 R0 FMAX	Reset the rotation
20 M30	End of program
21 END PGM 5 MM	

Example: Shoulder with recess



0 BEGIN PGM SHOULDER MM	
1 BLK FORM 0.1 Y X+0 Y-10 Z-35	Workpiece blank definition
2 BLK FORM 0.2 X+87 Y+10 Z+2	
3 TOOL CALL 12	Tool call
4 M140 MB MAX	Retract the tool
5 FUNCTION MODE TURN	Activate Turning mode
6 FUNCTION TURNDATA SPIN VCONST:ON VC:150	Constant surface speed
7 CYCL DEF 800 ADJUST XZ SYSTEM	Cycle definition: adapt the coordinate system
Q497=+0 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=0 ;INCLINED MACHINING	
Q531=+0 ;ANGLE OF INCIDENCE	
Q532=750 ;FEED RATE	
Q533=+0 ;PREFERRED DIRECTION	
Q535=3 ;ECCENTRIC TURNING	
Q536=0 ;ECCENTRIC W/O STOP	
8 M136	Feed rate in mm per revolution
9 L X+165 Y+0 R0 FMAX	Move to starting point in the plane
10 L Z+2 R0 FMAX M304	Set-up clearance, turning spindle on
11 CYCL DEF 812 SHOULDER, LONG. EXT.	Cycle definition: shoulder, longitudinal
Q215=+0 ;MACHINING OPERATION	
Q460=+2 ;SAFETY CLEARANCE	
Q491=+160 ;DIAMETER AT CONTOUR START	
Q492=+0 ;CONTOUR START IN Z	
Q493=+150 ;DIAMETER AT CONTOUR END	
Q494=-40 ;CONTOUR END IN Z	
Q495=+0 ;ANGLE OF CIRCUM. SURFACE	
Q501=+1 ;TYPE OF STARTING ELEMENT	
Q502=+2 ;SIZE OF STARTING ELEMENT	
Q500=+1 ;RADIUS OF CONTOUR EDGE	
Q496=+0 ;ANGLE OF FACE	

Q503=+1	;TYPE OF END ELEMENT	
Q504=+2	;SIZE OF END ELEMENT	
Q463=+2.5	;MAX. CUTTING DEPTH	
Q478=+0.25	;ROUGHING FEED RATE	
Q483=+0.4	;OVERSIZE FOR DIAMETER	
Q484=+0.2	;OVERSIZE IN Z	
Q505=+0.2	;FINISHING FEED RATE	
Q506=+0	;CONTOUR SMOOTHING	
12 CYCL CALL M8		Cycle call
13 M305		Turning spindle off
14 TOOL CALL 15		Tool call
15 M140 MB MAX		Retract the tool
16 FUNCTION TURNDATA SPIN VCONST:ON VC:100		Constant cutting speed
17 CYCL DEF 800 ADJUST XZ SYSTEM		Cycle definition: adapt the coordinate system
Q497=+0	;PRECESSION ANGLE	
Q498=+0	;REVERSE TOOL	
Q530=0	;INCLINED MACHINING	
Q531=+0	;ANGLE OF INCIDENCE	
Q532=750	;FEED RATE	
Q533=+0	;PREFERRED DIRECTION	
Q535=0	;ECCENTRIC TURNING	
Q536=+0	;ECCENTRIC W/O STOP	
18 L X+165 Y+0 R0 FMAX		Move to starting point in the plane
19 L Z+2 R0 FMAX M304		Set-up clearance, turning spindle on
20 CYCL DEF 862 EXPND. RECESS, RADL.		Cycle definition: recess
Q215=+0	;MACHINING OPERATION	
Q460=+2	;SAFETY CLEARANCE	
Q491=+150	;DIAMETER AT CONTOUR START	
Q492=-12	;CONTOUR START IN Z	
Q493=+142	;DIAMETER AT CONTOUR END	
Q494=-18	;CONTOUR END IN Z	
Q495=+0	;ANGLE OF SIDE	
Q501=+1	;TYPE OF STARTING ELEMENT	
Q502=+1	;SIZE OF STARTING ELEMENT	
Q500=+0	;RADIUS OF CONTOUR EDGE	
Q496=+0	;ANGLE OF SECOND SIDE	
Q503=+1	;TYPE OF END ELEMENT	
Q504=+1	;SIZE OF END ELEMENT	
Q478=+0.3	;ROUGHING FEED RATE	
Q483=+0.4	;OVERSIZE FOR DIAMETER	
Q484=+0.2	;OVERSIZE IN Z	
Q505=+0.15	;FINISHING FEED RATE	
Q463=+0	;LIMIT TO DEPTH	

21 CYCL CALL M8	Cycle call
22 M305	Turning spindle off
23 M137	Feed rate in mm per minute
24 M140 MB MAX	Retract the tool
25 FUNCTION MODE MILL	Activate Milling mode
26 M30	End of program
27 END PGM SHOULDER MM	

Example: turning, simultaneous finishing

The following NC program illustrates the use of Cycle 883 **TURNING SIMULTANEOUS FINISHING**.

Program run

- Tool call:turning tool
- Start turning mode
- Move to safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGINN PGM SIMULTAN MM	
1 BLK FORM CYLINDER Z D91 L40 DIST+0.5 DI+57.5	Workpiece blank definition
2 TOOL CALL "TURN"	Tool call
3 L Z+0 R0 FMAX M91	Retract the tool
4 FUNCTION MODE TURN	Activate turning mode
5 FUNCTION TURNDATA SPIN VCONST:ON VC:200 SMAX 800	Constant cutting speed
6 CYCL DEF 800 KOORD.-SYST. ANPASSEN	Cycle definition: adapt the coordinate system
Q497 =+0 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=+2 ;INCLINED MACHINING	
Q531=+1 ;ANGLE OF INCIDENCE	
Q532=MAX ;FEED RATE	
Q533=+1 ;PREFERRED DIRECTION	
Q535=+3 ;ECCENTRIC TURNING	
Q536=+0 ;ECCENTRIC W/O STOP	
7 M145	
8 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	Activate TCPM
9 CYCL DEF 14.0 KONTUR	Define the contour label
10 CYCL DEF 14.1 KONTURLABEL 2	
11 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING	Cycle definition: turning, simultaneous finishing
Q460=+2 ;SETUP CLEARANCE?	
Q499=+0 ;REVERSE CONTOUR	
Q558=-90 ;EXT:ANGLE CONT.START	
Q559=+90 ;CONTOUR END EXT ANGL	
Q505=+0.2 ;FINISHING FEED RATE	
Q556=-80 ;MIN. INCLINAT. ANGLE	
Q557=+60 ;MAX. INCLINAT. ANGLE	
Q555=+1 ;STEPPING ANGLE	
Q537=+0 ;INCID. ANGLE ACTIVE	
Q538=+0 ;INCLIN. ANGLE START	
Q539=+50 ;INCLINATN. ANGLE END	

Q565=+0 ;FINISHING ALLOW. D.	
Q566=+0 ;FINISHING ALLOW. Z	
Q567=+0 ;FINISH. ALLOW. CONT.	
12 L X+58 Y+0 R0 FMAX M303	Approach starting point
13 L Z+50 FMAX	Set-up clearance
14 CYCL CALL	Cycle call
15 L Z+50 FMAX	
16 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	Reset the coordinate system
17 M144	Cancel M145
18 FUNCTION MODE MILL	Activate milling mode
19 M30	End of program
20 LBL 2	
21 L X+58 Y+0 Z-1.5 RR	
22 L X+61 Z+0	
23 L X+88 Z+0	
24 L X+90 Z-1	
25 L X+90 Z-8	
26 L X+88 Z-10	
27 L X+88 Z-15	
28 L X+90 Z-17	
29 L X+90 Z-25	
30 RND R0.3	
31 L X+144 Z-25	
32 LBL 0	

15

Cycles: Grinding

15.1 Grinding cycles: general information

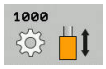
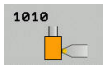
Overview

Proceed as follows to define grinding cycles:

- Press the **CYCL DEF** key

- Press the **GRINDING** soft key
- Select the cycle group, e.g. cycles for dressing
- Select the desired cycle, e.g. **DRESSING DIAMETER**.

The control offers the following cycles for grinding operations:

Soft key	Cycle group	Cycle	Page
	Reciprocating strokes		
		DEFINE RECIP. STROKE (Cycle 1000, DIN/ISO: G1000, option 156)	565
		START RECIP. STROKE (Cycle 1001, DIN/ISO: G1001, option 156)	568
		STOP RECIP. STROKE (Cycle 1002, DIN/ISO: G1002, option 156)	569
	Dressing		
		DRESSING DIAMETER (Cycle 1010, DIN/ISO: G1010, option 156)	570
		PROFILE DRESSING (Cycle 1015, DIN/ISO: G1015, option 156)	574
	Special cycles		
		ACTIVATE WHEEL EDGE (Cycle 1030 DIN/ISO: G1030, option 156)	578
		GRINDING WHL LENGTH COMPENSATION (Cycle 1032 DIN/ISO: G1032, option 156)	580
		GRINDING WHL RADIUS COMPENSATION (Cycle 1033 DIN/ISO: G1033, option 156)	582

General information on the grinding cycles

Jig grinding

Jig grinding means grinding of a 2-D contour. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin. Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

An NC program for grinding is structured as follows:

- **FUNCTION MODE MILL** – activate milling mode
- **TOOL CALL "grinding pin" Z S20000** – call grinding tool
- Cycle 1000 **DEFINE RECIP. STROKE** – define and start reciprocating stroke
- If required: Cycle 1001 **START RECIP. STROKE** – start reciprocating stroke
- E.g. call **LBL** "CONTOUR"
- E.g. Cycle 24 **SIDE FINISHING** – call cycle for finishing the contour
- E.g. **CYCL CALL** or call Cycle 24 with **M99**
- Cycle 1002 **STOP RECIP. STROKE** – stop reciprocating stroke

Dressing of grinding wheels

It is possible to dress or shape grinding tools in the machine. For this purpose, the grinding tool becomes the workpiece that is machined using a dressing tool. Identify dressing operations in your NC program with **FUNCTION DRESS BEGIN / END**. For dressing the diameter or profile of a grinding tool, you can use predefined cycles.

Refer to your machine manual!

The dressing mode can vary depending on the individual machine tool. Your machine tool builder may provide a simplified procedure for this purpose.

An NC program for dressing is structured as follows:

- **FUNCTION MODE MILL** – activate milling mode
- **TOOL CALL "grinding pin" Z S20000** – call grinding tool
- **L X... Y... Z..** – move the tool to be dressed to a position near the dressing tool
- **FUNCTION DRESS BEGIN** – if required, select the desired kinematics – activate the dressing mode
- Cycle 1030 **ACTIVATE WHEEL EDGE** – activate the wheel edge
- **TOOL CALL "dressing tool"** – call the dressing tool (no mechanical tool change)
- Call cycle 1010 **DRESSING DIAMETER** – call cycle for dressing the tool diameter
- **FUNCTION DRESS END** – deactivate the dressing mode

15.2 DEFINE RECIP. STROKE (Cycle 1000, DIN/ISO: G1000, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Using Cycle 1000 **DEFINE RECIP. STROKE**, you can define a reciprocating stroke in the tool axis and start reciprocating. This movement is executed as a superimposed movement. Thus, it is possible to execute any positioning block in parallel to the reciprocating stroke, even in the axis that is reciprocating. Once you started the reciprocating stroke, you can call a contour and start grinding.

- If you set **Q1004** to 0, no reciprocating stroke takes place. In this case, you only define the cycle. If required, call Cycle 1001 **START RECIP. STROKE** later to start the reciprocating stroke
- If you set **Q1004** to 1, reciprocating will start at the current position. Depending on the setting in **Q1002**, the control will start reciprocating the tool in the positive or negative direction first. This reciprocation movement will be superimposed on the programmed movements (X, Y, Z)

The following cycles can be called in combination with the reciprocating stroke:

- Cycle 24 **SIDE FINISHING**
- Cycle 25 **CONTOUR TRAIN**
- Cycles 25x **POCKETS/STUDS/SLOTS**
- Cycle 276 **THREE-D CONT. TRAIN**
- Cycle 274 **OCM FINISHING SIDE**

Please note while programming!

NOTICE

Danger of collision!

Collision monitoring (DCM) is not active during reciprocation movements. This means that movements that might cause collisions will not be prevented. There is a danger of collision!

- Verify the NC program by carefully executing it block by block

This cycle can be executed in the **FUNCTION MODE MILL** machining mode.

Cycle 1000 becomes effective immediately after its definition.

The control does not support mid-program startup while the reciprocating stroke is active.

The simulation of the superimposed movements is visible in the **Program run, single block** and **Program run, full sequence** operating modes.

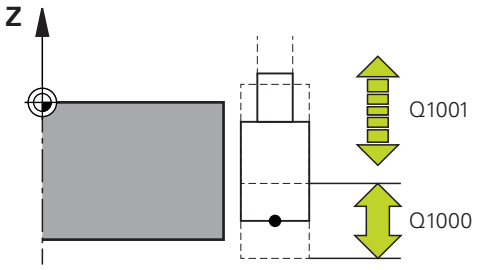
Stop the reciprocating movement when you no longer need it. To do so, use **M30** or Cycle 1002 **STOP RECIP. STROKE**. **M0** or **STOP** will not stop the reciprocating stroke.

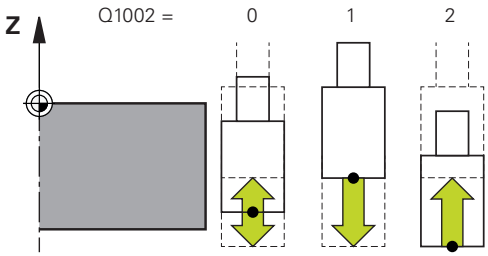
As long as the reciprocating stroke is active in a running NC program, you cannot change to the **Manual operation** or **Positioning w/ Manual Data Input** operating mode.

Reciprocating strokes can also be started in a tilted working plane. While the reciprocating stroke is active, however, you cannot change the orientation of the plane.

Cycle parameters

- **Q1000 Length of reciprocating stroke?**: Length of the reciprocation movement, parallel to the active tool axis. Input range: +0 to +9999.9999
- **Q1001 Feed rate for reciprocation?**: Feed rate of the tool during the reciprocating movement in mm/min. Input range: 0 to 999 999
- **Q1002 Type of reciprocation?**: Definition of the starting position. This defines the direction of the first reciprocating movement:
  0: The current position is considered as the center of the stroke movement. The control first moves the grinding tool by half the stroke in the negative direction and then continues the movement in the positive direction.
  –1: The current position is considered as the upper stroke limit. For the first stroke, the control traverses the grinding tool in the negative direction
  +1: The current position is considered as the lower stroke limit. For the first stroke, the control traverses the grinding tool in the positive direction
- **Q1004 Start reciprocating stroke?**: Definition of how this cycle will be effective:
  0: The reciprocating stroke is only defined and will be started at a later time
  +1: The reciprocating stroke is defined and the movement will start from the current position





Example

```
62 CYCL DEF 1000  DEFINE RECIP.
   STROKE
   Q1000=+0   ;RECIPROCATING STROKE
   Q1001=+999;RECIP. FEED RATE
   Q1002=+1   ;RECIPROCATION TYPE
   Q1004=+0   ;START RECIP. STROKE
```

15.3 START RECIP. STROKE (Cycle 1001, DIN/ISO: G1001, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Cycle 1001 **START RECIP. STROKE** starts a previously defined or stopped reciprocation movement. In an ongoing movement, this cycle has no effect.

Please note while programming!

This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
Cycle 1001 becomes effective immediately after its definition.
If you did not define a reciprocating stroke with Cycle 1000 **DEFINE RECIP. STROKE**, an error message will be displayed.

Cycle parameters

- Cycle 1001 does not have a cycle parameter. Finish the cycle input with the **END** key.

Example

```
62 CYCL DEF 1001  START RECIP.
   STROKE
```

15.4 STOP RECIP. STROKE (Cycle 1002, DIN/ISO: G1002, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Cycle 1002 **STOP RECIP. STROKE** stops the reciprocation movement. Depending on the setting in **Q1010**, the tool will stop immediately or traverse to its starting position.

Please note while programming!

This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
Cycle 1002 becomes effective immediately after its definition.

Cycle parameters

- **Q1005 Clear reciprocating stroke?**: Definition of how this cycle will be effective:
  0: The reciprocating stroke is only stopped and can be restarted at a later time, if necessary
  +1: The reciprocating stroke is stopped and the definition of the reciprocating stroke from Cycle 1000 is cleared
- **Q1010 Stop reciproc. immediately (1)?**: Definition of the stop position of the grinding tool:
  0: The stop position corresponds to the starting position
  +1: The stop position corresponds to the current position

Example

```
62 CYCL DEF 1002  STOP RECIP. STROKE
    Q1005=+0   ;CLEAR RECIP. STROKE
    Q1010=+0   ;RECIP.STROKE STOPPOS
```

15.5 DRESSING DIAMETER (Cycle 1010, DIN/ISO: G1010, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Using Cycle 1010 **DRESSING DIAMETER**, you can dress the diameter of your grinding wheel. Depending on the strategy, the control causes movements based on the wheel geometry. If you set the dressing strategy **Q1016** to 1 or 2, the return of the tool to the starting point does not run along the grinding wheel but via a retract path. Identify dressing operations in your NC program with **FUNCTION DRESS BEGIN / END**. In dressing mode, the control does not apply a tool radius compensation. Before starting the dressing cycle, activate a grinding wheel edge using Cycle 1030 **ACTIVATE WHEEL EDGE**. The control sets the datum for the dressing operation to this edge.

When you activate **FUNCTION DRESS BEGIN**, the grinding wheel is redefined as the workpiece and the dressing tool as the tool. This means that the tool axis moves in the opposite direction. In some cases, the other axes also move in the opposite direction. When you terminate the dressing mode with **FUNCTION DRESS END**, the grinding wheel is redefined as the tool.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin
1, 2, 5, 6	1, 3, 5, 7

Further information: "ACTIVATE WHEEL EDGE (Cycle 1030 DIN/ISO: G1030, option 156)", Page 578

Please note while programming!

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the kinematics is switched over. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Activate the **FUNCTION DRESS**dressing mode in the **Program run, single block** or **Program run, full sequence** operating mode only
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning is done in three axes at the same time. The control does not perform collision checking during this movement!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

If a dressing kinematics is active, the machine movements are effective in the opposite direction. There is a risk of collision when moving the axes!

- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

Cycle 1010 becomes effective immediately after its definition.

In dressing mode, no coordinate conversion cycles are permitted.

The control does not display dressing operations graphically! The machining times determined using the simulation do not correspond to the actual machining times.

The control does not support mid-program startup while dressing is active. If you jump to the first NC block after dressing using mid-program startup, the control will move the tool to the last position approached during dressing.

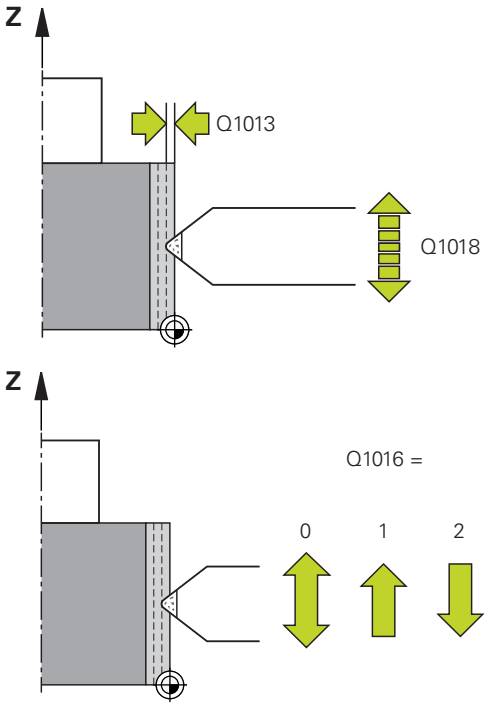
Each call of the dressing cycle increments a grinding wheel-specific counter. Dressing is only performed when the counter has reached the value from **Q1022**.

This cycle can only be run in dressing mode. The machine tool builder may already have programmed the switch-over in the cycle run.

Further information: User's Manual Conversational Programming

Cycle parameters

- **Q1013 Dressing amount?**: Value by which the control advances the tool for dressing on the radius path. Input range: 0 to 9.99999
- **Q1018 Feed rate for dressing?**: Traversing speed for dressing. Input range: 0 to 99 999
- **Q1016 Dressing strategy (0-2)?**: Definition of the traverse for dressing:
  0: Reciprocating, approach to and departure from the diameter are each done in both directions.
  1: Pull: In this case, the dressing tool is moved along the grinding wheel when approaching the active wheel edge
  2: Push: In this case, the dressing tool is moved along the grinding wheel when departing from the active wheel edge
- **Q1019 Number of dressing infeeds?**: Number of infeeds for the dressing operation. Input range: 1 to 999
- **Q1020 Number of idle strokes?**: Indicates how often the dressing tool will move above the grinding wheel without infeed when the dressing operation is finished. Input range: 0 to 99
- **Q1022 Dressing after number of calls?**: Number of called dressing cycles after which the control will execute the dressing operation. Input range: 0 to 99.
  0: The grinding tool will be dressed each time the dressing cycle is called
  >0 : Specify the number of dressing cycle calls after which the grinding wheel will be dressed
- **Q330 Tool number or tool name?** (optional): Number or name of the dressing tool. It is possible to take over the tool via soft key directly from the tool table.
  –1: The dressing tool has been activated before the dressing cycle
  Input range: –1 to +99 999.9
- **Q1011 Factor for cutting speed?** (optional): Factor specifying the ratio of the grinding wheel speed to the dressing tool cutting speed. Input range: –3 to +3
  0: The dressing tool is stationary
  >0: If you enter a value greater than zero, the dressing tool rotates with the grinding wheel at the contact point (opposite direction of rotation compared to the grinding wheel)
  <0: If you enter a value less than zero, the dressing tool rotates opposite to the grinding wheel at the contact point (same direction of rotation as the grinding wheel)



Example

```
62 CYCL DEF 1010  DRESSING DIAMETER
   Q1013=+0    ;DRESSING AMOUNT
   Q1018=+100  ;DRESSING FEED RATE
   Q1016=+1    ;DRESSING STRATEGY
   Q1019=+1    ;NUMBER INFEEDS
   Q1020=+0    ;IDLE STROKES
   Q1022=+0    ;COUNTER FOR DRESSING
   Q330=-1     ;TOOL
   Q1011=+0    ;FACTOR VC
```

15.6 PROFILE DRESSING (Cycle 1015, DIN/ISO: G1015, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Using Cycle 1015 **PROFILE DRESSING**, you can dress a defined profile of your grinding wheel. The profile must be defined in a separate NC program. This cycle is based on the grinding pin tool type. The activated wheel edge is used as the reference. The start and end points of the profile must be identical (closed path) and are located at a corresponding position on the selected wheel edge. You define the return path to the starting point in your profile program.

Identify dressing operations in your NC program with **FUNCTION DRESS BEGIN / END**. Depending on the profile program, the control either does or does not use tool radius compensation. Before starting the dressing cycle, activate a grinding wheel edge using Cycle 1030 **ACTIVATE WHEEL EDGE**. The control sets the datum for the dressing operation to this edge.

When you activate **FUNCTION DRESS BEGIN**, the grinding wheel is redefined as the workpiece and the dressing tool as the tool. This means that the tool axis moves in the opposite direction. In some cases, the other axes also move in the opposite direction. When you terminate the dressing mode with **FUNCTION DRESS END**, the grinding wheel is redefined as the tool.

This cycle supports the following wheel edges:

Grinding pin
1, 2, 5, 6

Further information: "ACTIVATE WHEEL EDGE (Cycle 1030 DIN/ISO: G1030, option 156)", Page 578

Please note while programming!

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the kinematics is switched over. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Activate the **FUNCTION DRESS**dressing mode in the **Program run, single block** or **Program run, full sequence** operating mode only
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning is done in three axes at the same time. The control does not perform collision checking during this movement!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

If a dressing kinematics is active, the machine movements are effective in the opposite direction. There is a risk of collision when moving the axes!

- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

Cycle 1015 becomes effective immediately after its definition.

In dressing mode, no coordinate conversion cycles are permitted.

The control does not display dressing operations graphically! The machining times determined using the simulation do not correspond to the actual machining times.

The control does not support mid-program startup while dressing is active. If you jump to the first NC block after dressing using mid-program startup, the control will move the tool to the last position approached during dressing.

The angle of infeed must be selected in a way that the programmed profile always remains within the grinding wheel edge. If this condition is not met, the dimensional accuracy of the grinding wheel is lost.

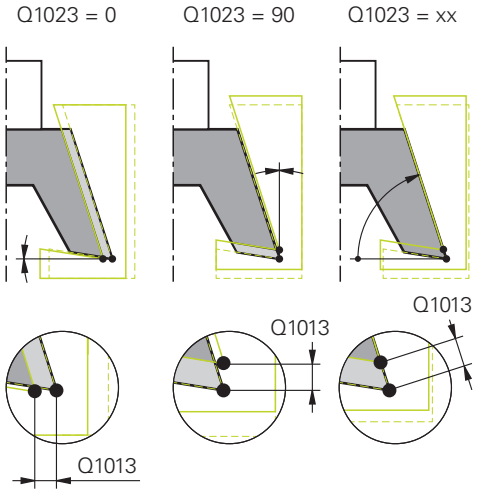
Each call of the dressing cycle increments a grinding wheel-specific counter. Dressing is only performed when the counter has reached the value from **Q1022**.

This cycle can only be run in dressing mode. The machine tool builder may already have programmed the switch-over in the cycle run.

Further information: User's Manual Conversational Programming

Cycle parameters

- **Q1013 Dressing amount?**: Value by which the control advances the tool for dressing on the radius path. Input range: 0 to 9.99999
- **Q1023 Infeed angle of profile program?**: Angle by which the programmed profile is offset into the grinding wheel. Input range: 0 to +90.
  0= Infeed only at the diameter in the direction of the principal axis
  +90= Infeed only in the direction of the tool axis
- **Q1018 Feed rate for dressing?**: Traversing speed for dressing. Input range: 0 to 99 999
- **Q1000 Name of the profile program?**: Name of the NC program used for programming the profile of the grinding tool for dressing.
- **Q1019 Number of dressing infeeds?**: Number of infeeds for the dressing operation. Input range: 1 to 999
- **Q1020 Number of idle strokes?**: Indicates how often the dressing tool will move above the grinding wheel without infeed when the dressing operation is finished. Input range: 0 to 99
- **Q1022 Dressing after number of calls?**: Number of called dressing cycles after which the control will execute the dressing operation. Input range: 0 to 99.
  0: The grinding tool will be dressed each time the dressing cycle is called
  >0 : Specify the number of dressing cycle calls after which the grinding wheel will be dressed
- **Q330 Tool number or tool name?** (optional): Number or name of the dressing tool. It is possible to take over the tool via soft key directly from the tool table.
  –1: The dressing tool has been activated before the dressing cycle
  Input range: –1 to +99 999.9
- **Q1011 Factor for cutting speed?** (optional): Factor specifying the ratio of the grinding wheel speed to the dressing tool cutting speed. Input range: –3 to +3
  0: The dressing tool is stationary
  >0: If you enter a value greater than zero, the dressing tool rotates with the grinding wheel at the contact point (opposite direction of rotation compared to the grinding wheel)
  <0: If you enter a value less than zero, the dressing tool rotates opposite to the grinding wheel at the contact point (same direction of rotation as the grinding wheel)



Example

```
62 CYCL DEF 1015 PROFILE DRESSING
    Q1013=+0    ;DRESSING AMOUNT
    Q1023=+0    ;ANGLE OF INFEED
    Q1018=+100  ;DRESSING FEED RATE
    QS1000=""   ;PROFILE PROGRAM
    Q1019=+1    ;NUMBER INFEEDS
    Q1020=+0    ;IDLE STROKES
    Q1022=+0    ;COUNTER FOR DRESSING
    Q330=-1     ;TOOL
    Q1011=+0    ;FACTOR VC
```

15.7 ACTIVATE WHEEL EDGE (Cycle 1030 DIN/ISO: G1030, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Use Cycle 1030 **ACTIVATE WHEEL EDGE** to activate the desired wheel edge. This means that you can change or update the reference point or reference edge. When dressing, you set the workpiece datum to the corresponding wheel edge with this cycle.

For this cycle, a distinction is made between grinding **(FUNCTION MODE MILL / TURN)** and dressing **(FUNCTION DRESS BEGIN / END)**.

Please note while programming!

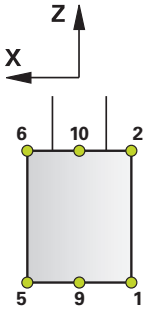
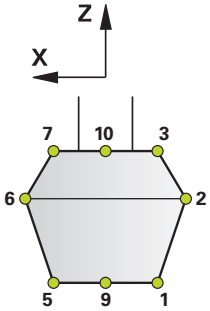
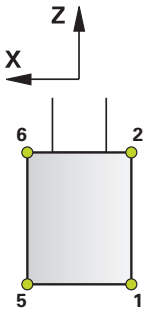
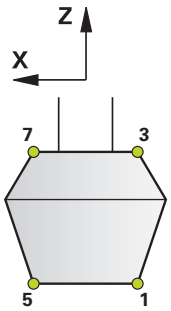
This cycle is only permitted in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes if a grinding tool has been activated.
Cycle 1030 becomes effective immediately after its definition.

Cycle parameters

- **Q1006 Edge of grinding wheel?**: Definition of the grinding tool edge.

Example

```
62 CYCL DEF 1030 ACTIVATE WHEEL
   EDGE
   Q1006=+9   ;WHEEL EDGE
```

Mode	Grinding pin	Special grinding pin
Grinding		
		
Dressing		
		

15.8 GRINDING WHL LENGTH COMPENSATION (Cycle 1032 DIN/ISO: G1032, option 156)

Cycle run

Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Using Cycle 1032 **GRINDING WHL LENGTH COMPENSATION**, you can define the overall length of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If no initial dressing has been carried out yet (checkbox for **INIT_D** not enabled), you can edit the basic data. Basic data affect both grinding and dressing.

If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Further information: User's Manual for Setup, Testing and Running NC programs

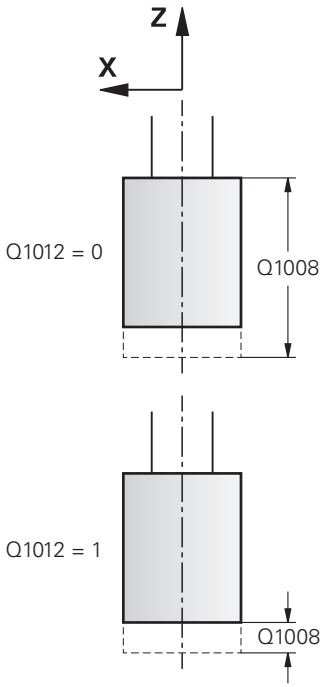
Please note while programming!

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Cycle 1032 becomes effective immediately after its definition.

Cycle parameters

- **Q1012 Compens. values (0=abs./1=inc.)?**: Defines how the length will be specified.
  0: Input of absolute length value
  1: Input of incremental length value
- **Q1008 Comp. value outside edge length?**: Dimension by which the tool length will be compensated, depending on **Q1012**, or entered as basic data. Input range: 0 to +999.99999.
  If you set **Q1012** to 0, you need to specify an absolute value for the length.
  If you set **Q1012** to 1, you need to specify an incremental value for the length.
- **Q330 Tool number or tool name?**: Number or name of the grinding tool. It is possible to take over the tool via soft key directly from the tool table. Input range: -1 to +99999.9



Example

```
62 CYCL DEF 1032  GRINDING WHL
   LENGTH COMPENSATION
   Q1012=+1   ;INCR. COMPENSATION
   Q1008=+0   ;COMP. OUTSIDE LENGTH
   Q330=-1    ;TOOL
```

15.9 GRINDING WHL RADIUS COMPENSATION (Cycle 1033 DIN/ISO: G1033, option 156)

Cycle run

Refer to your machine manual!

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

Using Cycle 1033 **GRINDING WHL RADIUS COMPENSATION**, you can define the radius of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If no initial dressing has been carried out yet (checkbox for **INIT_D** not enabled), you can edit the basic data. Basic data affect both grinding and dressing.

If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Further information: User's Manual for Setup, Testing and Running NC programs

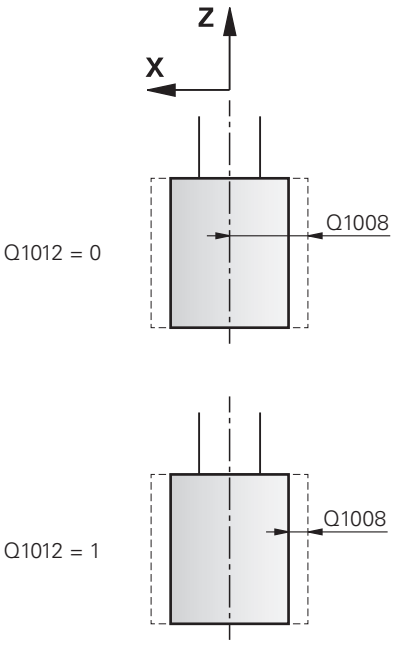
Please note while programming!

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Cycle 1033 becomes effective immediately after its definition.

Cycle parameters

- **Q1012 Compens. values (0=abs./1=inc.)?**: Defines how the radius will be specified.
  0: Input of absolute radius value
  1: Input of incremental radius value
- **Q1007 Compensation value for radius?**: Dimension by which the tool radius will be compensated, depending on **Q1012**. Input range: –999.99999 to +999.99999.
  If you set **Q1012** to 0, you need to specify an absolute value for the radius.
  If you set **Q1012** to 1, you need to specify an incremental value for the radius.
- **Q330 Tool number or tool name?**: Number or name of the grinding tool. It is possible to take over the tool via soft key directly from the tool table. Input range: -1 to +99999.9



Example

```
62 CYCL DEF 1033  GRINDING WHL
   RADIUS COMPENSATION
   Q1012=+1    ;INCR. COMPENSATION
   Q1007=+0    ;RADIUS COMPENSATION
   Q330=-1     ;TOOL
```

15.10 Programming examples

Example of grinding cycles

This programming example illustrates how to machine with a grinding tool.

The NC program uses the following cycles:

- Cycle 1000 **DEFINE RECIP. STROKE**
- Cycle 1002 **STOP RECIP. STROKE**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle 1000 **DEFINE RECIP. STROKE**
- Define Cycle 14 **CONTOUR GEOMETRY**
- Define Cycle 20 **CONTOUR DATA**
- Define and call Cycle 24 **SIDE FINISHING**
- Define Cycle 1002 **STOP RECIP. STROKE**

0 BEGIN PGM GRINDING_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL "GRINDING1" Z S20000	Tool call: grinding tool
5 L Z+30 R0 F1000	
6 CYCL DEF 1000 DEFINE RECIP. STROKE	Cycle definition: reciprocating stroke
Q1000=+13 ;RECIPROCATING STROKE	
Q1001=+25000 ;RECIP. FEED RATE	
Q1002=+1 ;RECIPROCATION TYPE	
Q1004=+1 ;START RECIP. STROKE	
7 CYCL DEF 14.0 CONTOUR GEOMETRY	
8 CYCL DEF 14.1 CONTOUR LABEL 1/2	
9 CYCL DEF 20 CONTOUR DATA	
Q1=-12 ;MILLING DEPTH	
Q2=+0.2 ;TOOL PATH OVERLAP	
Q3=+0.3 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=+2 ;SET-UP CLEARANCE	
Q7=+50 ;CLEARANCE HEIGHT	
Q8=+1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	

10 CYCL DEF 24 SIDE FINISHING	Cycle definition: side finishing
Q9=+1 ;ROTATIONAL DIRECTION	
Q10=-20 ;PLUNGING DEPTH	
Q11=+150 ;FEED RATE FOR PLNGNG	
Q12=+25 ;FEED RATE F. ROUGHNG	
Q14=+0.01 ;ALLOWANCE FOR SIDE	
11 CYCL CALL M13	Cycle call: side finishing
12 L Z+50 R0 FMAX	
13 CYCL DEF 1002 STOP RECIP. STROKE	Cycle definition: stop reciprocating stroke
Q1005=+1 ;CLEAR RECIP. STROKE	
Q1010=+0 ;RECIP.STROKE STOPPOS	
14 L X+100 Y+200 R0 FMAX	
15 L C+0 R0 FMAX M92	
16 STOP M30	End of program
17 LBL 1	Contour subprogram
18 L X+3 Y-23 RL	
19 L X-3	
20 CT X-9 Y-16	
21 CT X-7 Y-10	
22 CT X-7 Y+10	
23 CT X-9 Y+16	
24 CT X-3 Y+23	
25 L X+3	
26 CT X+9 Y+16	
27 CT X+7 Y+10	
28 CT X+7 Y-10	
29 CT X+9 Y-16	
30 CT X+3 Y-23	
31 LBL 0	
32 LBL 2	Contour subprogram
33 L X-25 Y-40 RR	
34 L Y+40	
35 L X+25	
36 L Y-40	
37 L X-25	
38 LBL 0	
39 END PGM GRINDING_CYCLE MM	

Example of dressing cycles

This programming example illustrates dressing mode.

The NC program uses the following cycles:

- Cycle 1030 **ACTIVATE WHEEL EDGE**
- Cycle 1010 **DRESSING DIAMETER**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle 1030 **ACTIVATE WHEEL EDGE**
- Tool call: Dressing tool (no mechanical tool change, but a switch-over for calculation)
- Cycle 1010 **DRESSING DIAMETER**
- Activate **FUNCTION DRESS END**

0 BEGIN PGM DRESS_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL "GRINDING1" Z S20000	Tool call: grinding tool
5 M140 MB MAX	
6 M3	
7 FUNCTION DRESS BEGIN	Activate dressing mode
8 CYCL DEF 1030 ACTIVATE WHEEL EDGE	Cycle definition: ACTIVATE WHEEL EDGE
Q1006=+5 ;WHEEL EDGE	
9 TOOL CALL 610	Tool call: dressing tool
10 L X+5 R0 F2000	
11 L Y+0 R0	
12 L Z-5 M8	
13 CYCL DEF 1010 DRESSING DIAMETER	Cycle definition: DRESSING DIAMETER
Q1013=+0 ;DRESSING AMOUNT	
Q1018=+300 ;DRESSING FEED RATE	
Q1016=+1 ;DRESSING STRATEGY	
Q1019=+2 ;NUMBER INFEEDS	
Q1020=+3 ;IDLE STROKES	
Q1022=+0 ;COUNTER FOR DRESSING	
Q330=-1 ;TOOL	
Q1011=+0 ;FACTOR VC	
14 FUNCTION DRESS END	Deactivate dressing mode
15 STOP M30	End of program
16 END PGM DRESS_CYCLE MM	

Example of a profile program

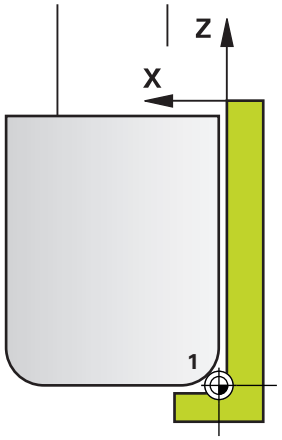
Grinding wheel edge no. 1

In this example, you program a profile of a grinding tool for dressing. The grinding tool has a radius at its outer edge.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)

Data to be used:

- Grinding wheel edge: 1
- Set-up clearance: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm



0 BEGIN PGM PROFIL MM	
1 L X-5 Z-5 R0 FMAX	Move to initial position
2 L Z+45 RL FMAX	Move to starting position
3 L X+0 FQ1018	Q1018 = Feed rate for dressing
4 L Z+0 FQ1018	Move to radius edge
5 RND R+2 FQ1018	Rounding
6 L X+6 FQ1018	Move to end position X
7 L Z-5 FQ1018	Move to end position Z
8 L X-5 Z-5 R0 FMAX	Move to initial position
9 END PGM PROFIL MM	

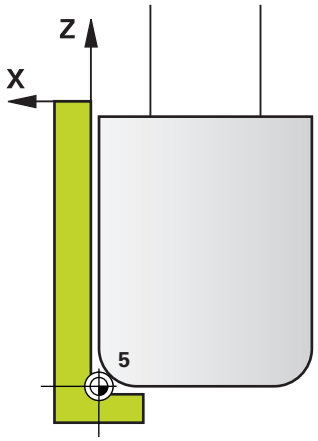
Grinding wheel edge no. 5

In this example, you program a profile of a grinding tool for dressing. The grinding tool has a radius at its outer edge.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)

Data to be used:

- Grinding wheel edge: 5
- Set-up clearance: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm



0 BEGIN PGM PROFIL MM	
1 L X+5 Z-5 R0 FMAX	Move to initial position
2 L Z+45 RR FMAX	Move to starting position
3 L X+0 FQ1018	Q1018 = Feed rate for dressing
4 L Z+0 FQ1018	Move to radius edge
5 RND R+2 FQ1018	Rounding
6 L X-6 FQ1018	Move to end position X
7 L Z-5 FQ1018	Move to end position Z
8 L X+5 Z-5 R0 FMAX	Move to initial position
9 END PGM PROFIL MM	

16

Using Touch Probe Cycles

16.1 General information about touch probe cycles

The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

Touch probe functions deactivate the **Global Program Settings** temporarily.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Method of function

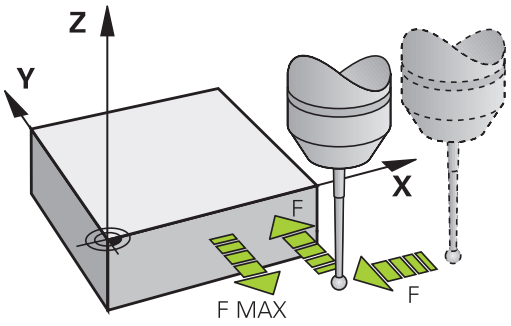
Whenever the control runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further information: "Before You Start Working with Touch Probe Cycles", Page 593

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: 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 control displays an error message (distance: **DIST** from touch probe table).



Consideration of a basic rotation in the Manual operation mode

During probing, the control considers an active basic rotation and approaches the workpiece at an angle.

Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

In the **Manual operation** and **Electronic handwheel** modes, the control provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Set presets

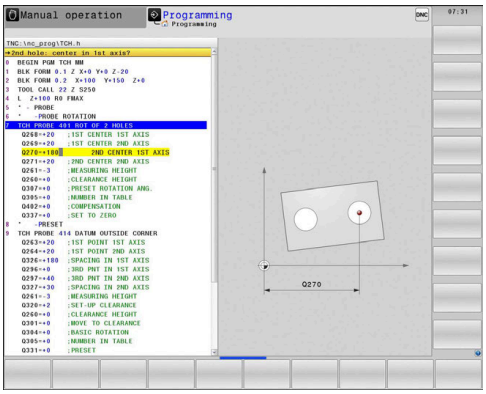
Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual Operation and Electronic handwheel modes, the control provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Presetting
- Automatic workpiece inspection
- Automatic tool measurement

You can program the touch probe cycles in the **Programming** operating mode via the **TOUCH PROBE** key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with the same function that the control requires 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 control shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted (see figure right).



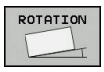
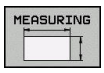
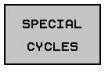
Defining the touch probe cycle in the Programming mode of operation

Proceed as follows:

- Press the **TOUCH PROBE** key.

- Select the desired probing cycle group, e.g. presetting
- Cycles for automatic tool measurement are available only if your machine has been prepared for them.

- Select a cycle, e.g. presetting at pocket center
- The control opens the programming dialog and prompts you for all required input values. At the same time, a graphic of the input parameters is displayed in the right screen half. The parameter you are prompted for in the dialog is highlighted.
- Enter all parameters required by the control
- Confirm each input with the **ENT** key
- The control closes the dialog when all required data has been entered.

Soft key	Measuring cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	600
	Cycles for automatic workpiece presetting	648
	Cycles for automatic workpiece inspection	706
	Special cycles	754
	Calibrate TS	766
	Kinematics	803
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	842
	Camera-based monitoring (option 136 VSC)	782

NC blocks

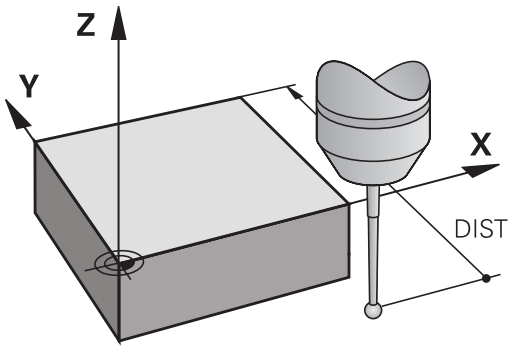
```
5 TCH PROBE 410  PRESET INSIDE RECTAN.
   Q321=+50   ;CENTER IN 1ST AXIS
   Q322=+50   ;CENTER IN 2ND AXIS
   Q323=60    ;FIRST SIDE LENGTH
   Q324=20    ;2ND SIDE LENGTH
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE
   Q305=10    ;NUMBER IN TABLE
   Q331=+0    ;PRESET
   Q332=+0    ;PRESET
   Q303=+1    ;MEAS. VALUE TRANSFER
   Q381=1     ;PROBE IN TS AXIS
   Q382=+85   ;1ST CO. FOR TS AXIS
   Q383=+50   ;2ND CO. FOR TS AXIS
   Q384=+0    ;3RD CO. FOR TS AXIS
   Q333=+0    ;PRESET
```

16.2 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.

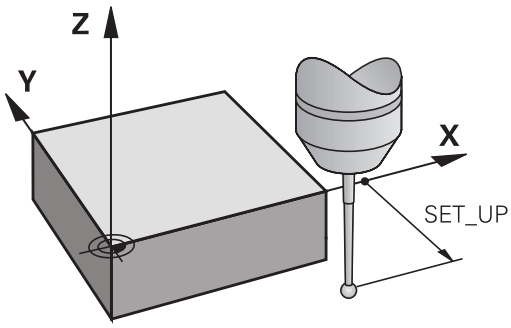
Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the range defined in **DIST**, the control will issue 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 control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to **SET_UP**.



Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: F in touch probe table

In **F**, you define the feed rate at which the control is to probe the workpiece.

F can never be greater than set in the optional **maxTouchFeed** machine parameter (no. 122602).

The feed rate potentiometer may be effective with touch probe cycles. The machine tool builder defines the required settings. (the parameter **overrideForMeasure** (No. 122604) must be appropriately configured.)

Touch trigger probe, rapid traverse for positioning: FMAX

In **FMAX**, you define the feed rate at which the control pre-positions the touch probe and positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table

In **F_PREPOS**, you define whether the control is to position the touch probe at the feed rate defined in FMAX or at rapid traverse.

- Input value = **FMAX_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX_MACHINE**: Pre-position at rapid traverse

Executing touch probe cycles

All touch probe cycles are DEF-active. This means that the control runs the cycle automatically as soon as it executes the cycle definition in the program run.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 1400 to 1499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **8 MIRROR IMAGE**, Cycles **11 SCALING** and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand.

Depending on the setting of the optional **chkTiltingAxes** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

Touch probe cycles 408 to 419 and 1400 to 1499 may be run even if a basic rotation is active. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT after the measuring cycle.

Touch probe cycles numbered 400 to 499 or 1400 to 1499 position the touch probe according to the following positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control 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.

16.3 Touch-probe table

General information

Various data are stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.

> You can also view and edit the data of the touch probe table in the expanded tool management (option 93).

Editing touch probe tables

Proceed as follows:

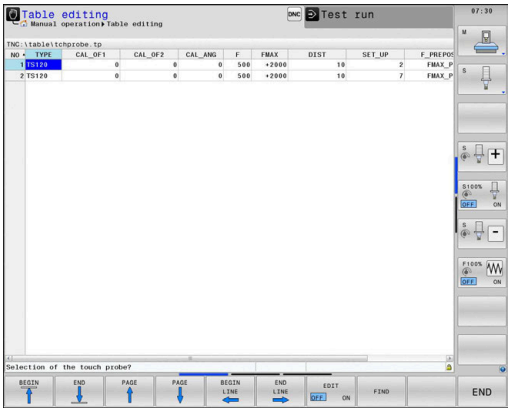
- Press the **Manual operation** key

- Press the **TOUCH PROBE** soft key

> The control displays additional soft keys.

- Press the **TCH PROBE TABLE** soft key

- Set the **EDIT** soft key to **ON**.
- Using the arrow keys, select the desired setting.
- Perform desired changes.
- Exit the touch probe table: Press the **END** soft key.



Touch probe data

Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool entries	–
TYPE	Selection of the touch probe used	**Selection of the touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis in the secondary axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the control will probe the workpiece **F** can never be greater than set in the optional **maxTouch-Feed** machine parameter (no. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioned and is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within this defined value, the control will issue an error message.	**Maximum measuring range? [mm]**
SET_UP	In **set_up** you define how far from the defined or calculated touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the **SET_UP** machine parameter.	**Set-up clearance? [mm]**
F_PREPOS	Defining speed for pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-position at rapid? ENT/ NOENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probing process. In this way, the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Probe oriented? Yes=ENT/ No=NOENT**
SERIAL	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface.	**Serial number?**

Abbr.	Inputs	Dialog
REACTION	As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. The entry defines how the control is to react to the resetting of the ready signal ■ **NCSTOP**: The NC program will be interrupted ■ **EMERGSTOP**: Emergency stop, quick braking of the axes	**Reaction?**

With **TS 642** touch probes, you can select between **TS642-3** and **TS642-6** in the **TYPE** column. Values 3 and 6 correspond to the switch settings in the battery case of the touch probe.

- **3**: Activation of the touch probe via a switch in the taper shank. Do not use this mode. It is not supported by HEIDENHAIN controls yet.
- **6**: Activation of the touch probe via an infrared signal. Select this mode.

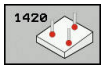
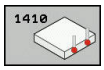
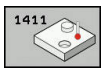
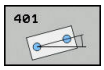
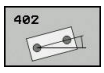
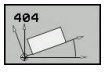
17

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

17.1 Overview

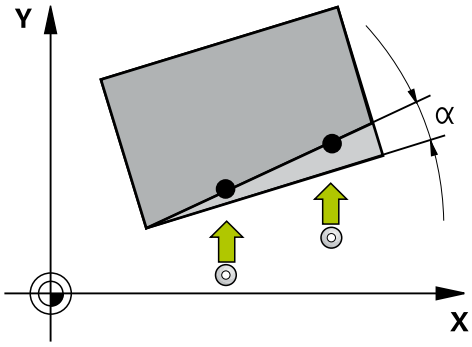
The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Cycle	Page
	1420 PROBING IN PLANE Automatic measurement using three points. Compensation via basic rotation.	609
	1410 PROBING ON EDGE Automatic measurement using two points. Compensation via basic rotation or a rotation of the rotary table.	613
	1411 PROBING TWO CIRCLES Automatic measurement using two cylindrical holes or studs. Compensation via basic rotation or a rotation of the rotary table.	617
	400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.	623
	401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.	626
	402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.	630
	403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.	635
	405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.	640
	404 SET BASIC ROTATION Setting any basic rotation.	644

17.2 Touch probe cycles 14xx: fundamentals

Characteristics common to the 14xx touch probe cycles for measuring rotations



Rotations can be determined using the following three cycles:

- 1410 **PROBING ON EDGE**
- 1411 **PROBING TWO CIRCLES**
- 1420 **PROBING IN PLANE**

These cycles include the following functionality:

- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3-D calibration
- Rotation and position are measured simultaneously

The probing positions reference the programmed nominal positions in the I-CS.

See the drawing for these nominal positions.

Before a cycle definition you must program a tool call to define the touch-probe axis.

Explanation of terms

Designation	Short description
Nominal position	Position in the drawing, e.g. position of a hole
Nominal dimension	Dimension in the drawing, e.g. hole diameter
Actual position	Measured position, e.g. position of a hole
Actual dimension	Measured dimension, e.g. hole diameter
I-CS	I-CS: **Input Coordinate System**
W-CS	W-CS: **Workpiece Coordinate System**
Object	Object to be probed: circle, stud, plane, edge

Evaluation – preset:

- If you want to probe objects in a consistent machining plane or probe objects while TCPM is active, you can program any required shifts as basic transformations in the preset table.
- In the basic transformations, rotations can be programmed as basic rotations or as axial offsets from the first rotary table axis, seen from the workpiece.

When probing, existing 3-D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.

If you want to use not only the measured rotation, but also a measured position, make sure to probe the surface perpendicularly, if possible. The larger the angular error and the bigger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.

Logging:

The determined results are recorded in the **TCHPRAUTO.html** file and stored in the Q parameters programmed for this cycle. The measured deviations are the differences between the measured actual values and the mean tolerance value. If no tolerance has been specified, they refer to the nominal dimension.

Semi-automatic mode

If the probing positions relative to the current datum are unknown, you can execute the cycle in semi-automatic mode. In this mode, you can determine the starting position by manually pre-positioning before performing the probing operation on the desired object.

For this purpose, precede the value for the required nominal position with **"?"**. This can be done via the **ENTER TEXT** soft key. Depending on the object, you need to define the nominal positions that determine the probing direction, see "Examples".

Cycle run:

1 The cycle interrupts the NC program.

2 A dialog window opens

Proceed as follows:

- Use the axis-direction keys to pre-position the touch probe at the desired point
- Alternatively, use the handwheel for pre-positioning
- If required, change the probing conditions, such as the probing direction.
- Press **NC start**

> If you programmed value 1 or 2 for **Q1125** Traverse to clearance height, then the control will open a pop-up window, explaining that the Traverse to clearance height mode cannot be used here.

- While the pop-up window is still open, use the axis keys to move to a safe position
- Press **NC start**

> Program execution is resumed.

NOTICE

Danger of collision!

The control will ignore the programmed values 1 or 2 for Traverse to clearance height when running in semi-automatic mode. Depending on the position of the touch probe, there is danger of collision.

- In semi-automatic mode, traverse to clearance height after each probing operation.

See the drawing for these nominal positions.

Semi-automatic mode can only be executed in the machine operating modes, i.e. not in the Test run operating mode.

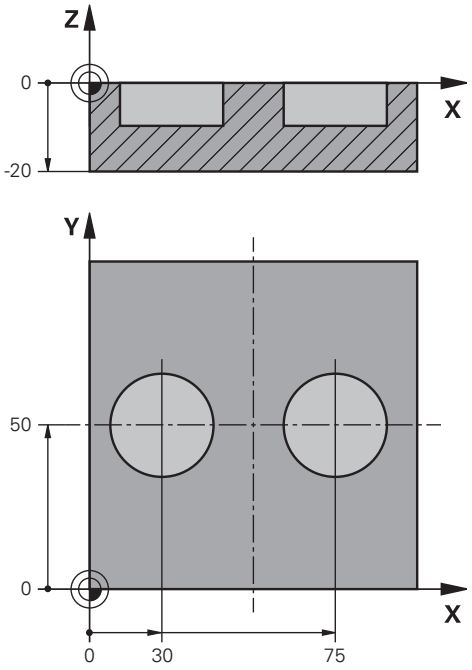
If you did not define a nominal position for a touch point in any direction, the control generates an error message.

If you did not define a nominal position for a single direction, the control will capture the actual position after probing the object. This means that the measured actual position will subsequently be applied as the nominal position. Consequentially, there is no deviation for this position and thus no position compensation.

Examples

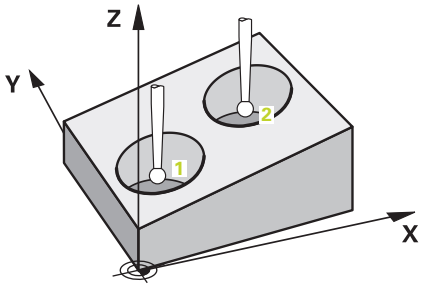
Important: Specify the **nominal positions** from the drawing!

In the following three examples, the nominal positions from this drawing will be used.



Hole

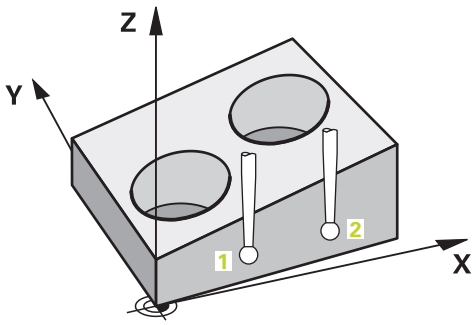
In this example, you will align two holes. Probing is done in the X axis (principal axis) and in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.



5 TCH PROBE 1411 PROBING TWO CIRCLES		Define the cycle
QS1100= "?30"	**;1ST POINT REF AXIS**	Nominal position 1 of principal axis exists, position of the workpiece unknown
QS1101= "?50"	**;1ST POINT MINOR AXIS**	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?"	**;1ST POINT TOOL AXIS**	Nominal position 1 of tool axis unknown
Q1116=+10	**;DIAMETER 1**	Diameter at 1st position
QS1103= "?75"	**;2ND POINT REF AXIS**	Nominal position 2 of principal axis exists, position of the workpiece unknown
QS1104= "?50"	**;2ND POINT MINOR AXIS**	Nominal position 2 of secondary axis exists, position of workpiece unknown
QS1105= "?"	**;2ND POINT TOOL AXIS**	Nominal position 2 of tool axis unknown
Q1117=+10	**;DIAMETER 2**	Diameter at 2nd position
Q1115=+0	**;GEOMETRY TYPE**	Geometry type: two holes
...	**;**	

Edge

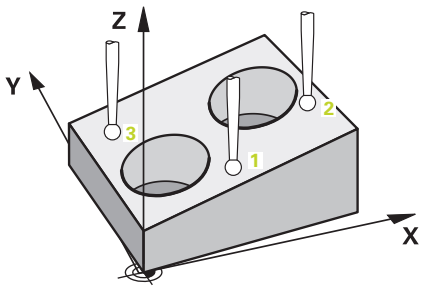
In this example, you will align an edge. Probing is on done in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for this axis! Nominal positions for the X axis (principal axis) and for the Z axis (tool axis) are not required because you will not measure in these directions.



5 TCH PROBE 1410 PROBING ON EDGE	Define the cycle
QS1100= "?" ;1ST POINT REF AXIS	Nominal position 1 of principal axis unknown
QS1101= "?0" ;1ST POINT MINOR AXIS	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?" ;1ST POINT TOOL AXIS	Nominal position 1 of tool axis unknown
QS1103= "?" ;2ND POINT REF AXIS	Nominal position 2 of principal axis unknown
QS1104= "?0" ;2ND POINT MINOR AXIS	Nominal position 2 of secondary axis exists, position of workpiece unknown
QS1105= "?" ;2ND POINT TOOL AXIS	Nominal position 2 of tool axis unknown
Q372=+2 ;PROBING DIRECTION	Probing direction Y+
... ;	

Plane

In this example, you will align a plane. In this case, it is mandatory to define all three nominal positions. For angle calculations, it is important that all three axes are taken into account when probing.



5 TCH PROBE 1420 PROBING IN PLANE	Define the cycle
QS1100= "?50" ;1ST POINT REF AXIS	Nominal position 1 of principal axis exists, position of the workpiece unknown
QS1101= "?10" ;1ST POINT MINOR AXIS	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?0" ;1ST POINT TOOL AXIS	Nominal position 1 of tool axis exists, position of the workpiece unknown
QS1103= "?80" ;2ND POINT REF AXIS	Nominal position 2 of principal axis exists, position of the workpiece unknown
QS1104= "?50" ;2ND POINT MINOR AXIS	Nominal position 2 of secondary axis exists, position of workpiece unknown
QS1105= "?0" ;2ND POINT TOOL AXIS	Nominal position 2 of tool axis exists, position of the workpiece unknown
QS1106= "?20" ;3RD POINT REF AXIS	Nominal position 3 of principal axis exists, position of the workpiece unknown
QS1107= "?80" ;3RD POINT MINOR AXIS	Nominal position 3 of secondary axis exists, position of workpiece unknown
QS1108= "?0" ;3RD POINT TOOL AXIS	Nominal position 3 of tool axis exists, position of the workpiece unknown
Q372=-3 ;PROBING DIRECTION	Probing direction Z-
... ;	

Evaluation of tolerances

Optionally, the control can monitor the cycles for tolerances. This includes monitoring the position and size of an object.

If you added a tolerance to a dimension, the control will monitor it and set an error status in the **Q183** return parameter if the tolerance is not met. Tolerance monitoring and the status always refer to the situation during probing. The preset will be corrected only afterwards, if required.

Cycle run:

- With an error reaction parameter set to **Q309**=1, the control will check for scrap and rework. If you defined **Q309**=2, the control will only check for scrap.
- If the actual position determined by probing is erroneous, the control will interrupt execution of the NC program. A dialog window opens, showing all nominal and actual dimensions of the object
- You can then decide whether to continue machining or abort the NC program. To resume NC program operation, press **NC start**. To abort the program, press the **CANCEL** soft key

Please keep in mind that the touch probe cycles return the deviations with respect to the mean tolerance value in Q parameters **Q98x** and **Q99x**. Thus, these values equal the compensation values used by the cycle if input parameters **Q1120** and **Q1121** have been set accordingly. If no automatic evaluation has been programmed, the control will save the values with respect to the mean tolerance in the programmed Q parameters. You can further process them.

5 TCH PROBE 1410 PROBING TWO CIRCLES	Define the cycle
Q1100=+50 ;1ST POINT REF AXIS	Nominal position 1 of reference axis
Q1101= +50 ;1ST POINT MINOR AXIS	Nominal position 1 of minor axis
Q1102= -5 ;1ST POINT TOOL AXIS	Nominal position 1 of tool axis
QS1116="+9-1-0.5" ;DIAMETER 1	Diameter 1, tolerance specified
Q1103= +80 ;2ND POINT REF AXIS	Nominal position 2 of reference axis
Q1104=+60 ;2ND POINT MINOR AXIS	Nominal position 2 of minor axis
QS1105= -5 ;2ND POINT TOOL AXIS	Nominal position 2 of tool axis
QS1117="+9-1-0,5" ;DIAMETER 2	Diameter 2, tolerance specified
... ;	
Q309=2 ;ERROR REACTION	
... ;	

Transferring the actual position

You can determine the actual position in advance and define it as the actual position for the touch probe cycle. Then, both the nominal position and the actual position will be transferred. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

For this purpose, enter **"@"**after the value for the required nominal position. This can be done via the **ENTER TEXT** soft key. Enter the actual position after the "**@**".

If you program @, no probing will be carried out. The control only accounts for the actual and nominal positions.

You must define the actual position for all three axes: principal axis, secondary axis, and tool axis. If you define only one axis with its actual position, an error message will be generated.

Actual positions can also be defined with Q parameters **Q1900-Q1999**.

Example:

This feature allows you to do the following:

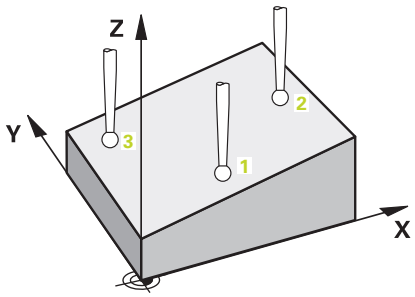
- Determine a circular pattern based on multiple different objects
- Align a gear wheel via its center and the position of a tooth

5 TCH PROBE 1410 PROBING ON EDGE	
QS1100= "10+0.02@10.0123"	
;1ST POINT REF AXIS	Nominal position 1 of the principal axis with tolerance monitoring and actual position
QS1101="50@50.0321"	
;1ST POINT MINOR AXIS	Nominal position 1 of the secondary axis with tolerance monitoring and actual position
QS1102= "-10-0.2+0.02@Q1900"	
;1ST POINT TOOL AXIS	Nominal position 1 of the tool axis with tolerance monitoring and actual position
... ;	

17.3 PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420)

Cycle run

Touch probe cycle 1420 finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.



1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic ("Executing touch probe cycles"), at the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.

2 If you programmed Traverse to clearance height, the touch probe returns to clearance height (depending on the setting in **Q1125**). It then moves in the working plane to touch point **2** to measure there the actual value of the second touch point in the plane.

3 The touch probe returns to the clearance height (depends on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.

4 Finally the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:

Parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the principal, secondary, and tool axes
Q953 to **Q955**	Measured position 2 in the principal, secondary, and tool axes
Q956 to **Q958**	Measured position 3 in the principal, secondary, and tool axes
Q961 to **Q963**	Measured spatial angle SPA, SPB, and SPC in the W-CS
Q980 to **Q982**	Deviation 1 of the positions
Q983 to **Q985**	Deviation 2 of the positions
Q986 to **Q988**	Deviation 3 of the positions
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

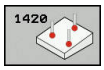
The control can only calculate the angular values if the three touch points are not positioned on a straight line.

The nominal spatial angle results from the defined nominal positions. The control saves the calculated spatial angle in parameters **Q961** to **Q963**. The control transfers the difference between the measured spatial angle and the nominal spatial angle to the basic rotation in 3-D of the preset table.

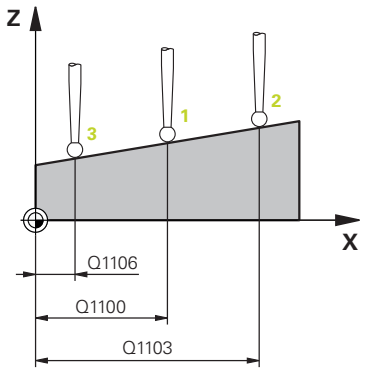
Aligning the rotary table axes:

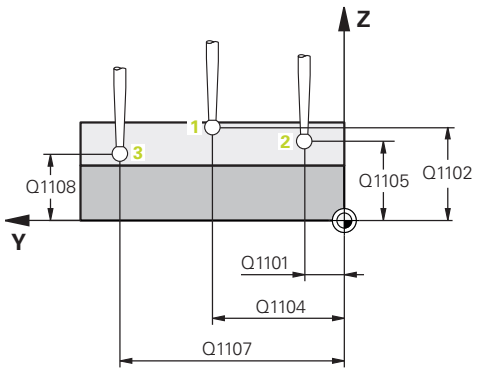
- Alignment with rotary table axes is only possible if two rotary table axes have been defined in the kinematics.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible to align the rotary table axes without defining evaluation of the rotation.

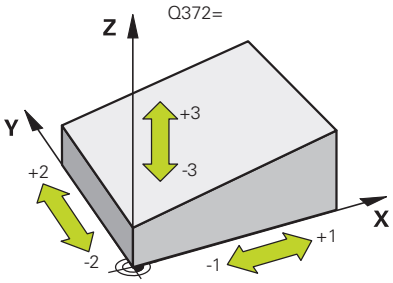
Cycle parameters

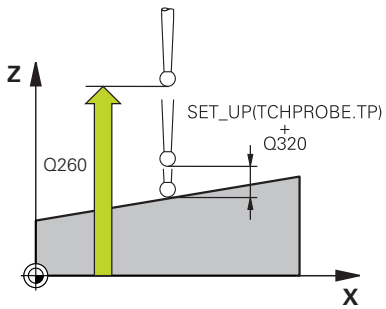


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal position of the first touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal position of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position of the second touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1106 3rd noml. position of ref axis?** (absolute): Nominal position of the third touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1107 3rd noml. position minor axis?** (absolute): Nominal position of the third touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1108 3rd nominal position tool axis?** (absolute): Nominal position of the third touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input range: –3 to +3
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999









- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the touch points:
  –1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each object
  2: Move to clearance height before and after moving to each touch point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the program run.
  In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.
- **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
  0: Maintain the current tilting axes position
  1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the ball tip (TURN)
- **Q1120 Transfer position?**: Define which touch point will be used to correct the current preset:
  0: No correction
  1: Correction based on the 1st touch point
  2: Correction based on the 2nd touch point
  3: Correction based on the 3rd touch point
  4: Correction based on the averaged touch point position
- **Q1121 Confirm basic rotation?**: Define whether the control is to transfer the determined inclination as the basic rotation:
  0: No basic rotation
  1: Set basic rotation: The control saves the basic rotation.

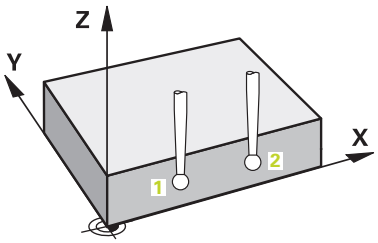
Example

```
5 TCH PROBE 1420  PROBING IN PLANE
  Q1100=+0    ;1ST POINT REF AXIS
  Q1101=+0    ;1ST POINT MINOR AXIS
  Q1102=+0    ;1ST POINT TOOL AXIS
  Q1103=+0    ;2ND POINT REF AXIS
  Q1104=+0    ;2ND POINT MINOR AXIS
  Q1105=+0    ;2ND POINT TOOL AXIS
  Q1106=+0    ;3RD POINT REF AXIS
  Q1107=+0    ;3RD POINT MINOR AXIS
  Q1108=+0    ;3RD POINT MINOR AXIS
  Q372=+1     ;PROBING DIRECTION
  Q320=+0     ;SET-UP CLEARANCE
  Q260=+100   ;CLEARANCE HEIGHT
  Q1125=+2    ;CLEAR. HEIGHT MODE
  Q309=+0     ;ERROR REACTION
  Q1126=+0    ;ALIGN ROTARY AXIS
  Q1120=+0    ;TRANSER POSITION
  Q1121=+0    ;CONFIRM ROTATION
```

17.4 PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410)

Cycle run

Touch probe cycle 1410 determines the misalignment of a workpiece by measuring two points located on an edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic ("Executing touch probe cycles"), at the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe in the direction opposite to the direction of probing

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 The touch probe then moves to the next touch point **2** and probes again.

4 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:

Parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the principal, secondary, and tool axes
Q953 to **Q955**	Measured position 2 in the principal, secondary, and tool axes
Q964	Measured angle of rotation in the I-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q980 to **Q982**	Deviation 1 of the positions
Q983 to **Q985**	Deviation 2 of the positions
Q994	Measured angle deviation in the I-CS
Q995	Measured angle deviation in the coordinate system of the rotary table
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

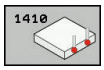
- Always move to clearance height between objects or touch points

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

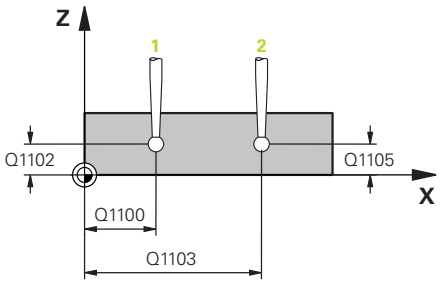
Aligning the rotary table axes:

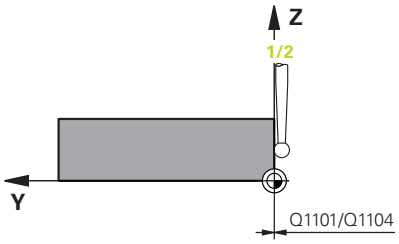
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible to align the rotary table axes and activate the basic rotation at the same time

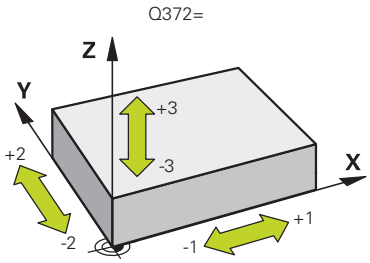
Cycle parameters

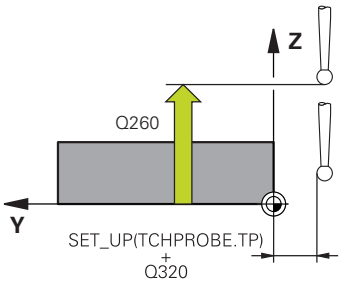


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal position of the first touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal position of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position of the second touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input range: –3 to +3
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999









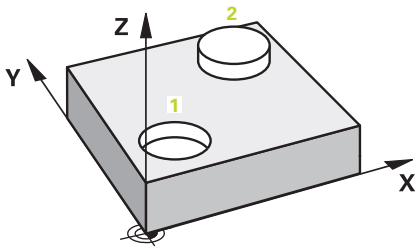
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the touch points:
  –1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each object
  2: Move to clearance height before and after moving to each touch point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the program run.
  In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.
- **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
  0: Maintain the current tilting axes position
  1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the ball tip (TURN)
- **Q1120 Transfer position?**: Define which touch point will be used to correct the current preset:
  0: No correction
  1: Correction based on the 1st touch point
  2: Correction based on the 2nd touch point
  3: Correction based on the averaged touch point position
- **Q1121 CONFIRM ROTATION?**: Define whether the control is to transfer the determined inclination as the basic rotation:
  0: No basic rotation
  1: Set basic rotation: The control saves the basic rotation
  2: Perform rotation of the rotary table: The control will make an entry in the corresponding **Offset** column of the preset table

Example

```
5 TCH PROBE 1410  PROBING ON EDGE
   Q1100=+0   ;1ST POINT REF AXIS
   Q1101=+0   ;1ST POINT MINOR AXIS
   Q1102=+0   ;1ST POINT TOOL AXIS
   Q1103=+0   ;2ND POINT REF AXIS
   Q1104=+0   ;2ND POINT MINOR AXIS
   Q1105=+0   ;2ND POINT TOOL AXIS
   Q372=+1    ;PROBING DIRECTION
   Q320=+0    ;SET-UP CLEARANCE
   Q260=+100  ;CLEARANCE HEIGHT
   Q1125=+2   ;CLEAR. HEIGHT MODE
   Q309=+0    ;ERROR REACTION
   Q1126=+0   ;ALIGN ROTARY AXIS
   Q1120=+0   ;TRANSER POSITION
   Q1121=+0   ;CONFIRM ROTATION
```

17.5 PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411)

Cycle run



Touch probe cycle 1411 captures the center points of two holes or cylindrical studs and calculates a straight line connecting these center points. The cycle determines the rotation in the working plane based on the difference between the measured angle and the nominal angle.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic ("Executing touch probe cycles"), at the programmed center point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.

2 Then the probe moves to the entered measuring height and probes (depending on the number of probes in **Q423**) the first hole or stud center point.

3 The touch probe returns to the clearance height and then to the position entered as center of the second hole or second stud **2**.

4 Then the control moves the probe to the entered measuring height and probes (depending on the number of probes in **Q423**) the second hole or stud center point.

5 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:

Parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the principal, secondary, and tool axes
Q953 to **Q955**	Measured position 2 in the principal, secondary, and tool axes
Q964	Measured angle of rotation in the I-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q966 to **Q967**	Measured first and second diameters
Q980 to **Q982**	Deviation 1 of the positions
Q983 to **Q985**	Deviation 2 of the positions
Q994	Measured angle deviation in the I-CS
Q995	Measured angle deviation in the coordinate system of the rotary table
Q996 to **Q997**	Measurement error of the first and second diameters
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

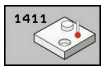
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the hole is so small that the programmed set-up clearance cannot be met, a dialog opens. It shows the nominal dimension of the hole, the calibrated ball-tip radius, and the achievable set-up clearance.
Either confirm the dialog with **NC start** or cancel the process via a soft key. If you confirm with **NC start**, the effective set-up clearance will be reduced to the displayed value, but only for this object.

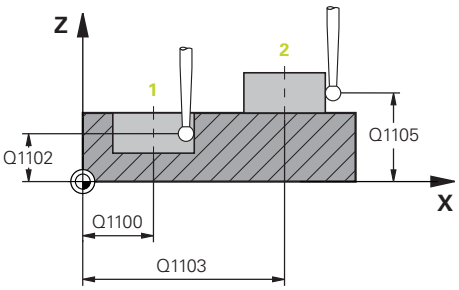
Aligning the rotary table axes:

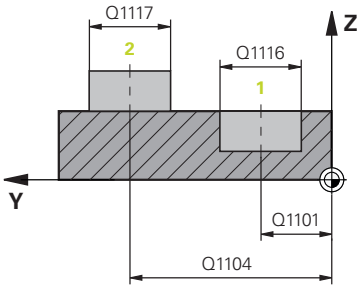
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible to align the rotary table axes and activate the basic rotation at the same time

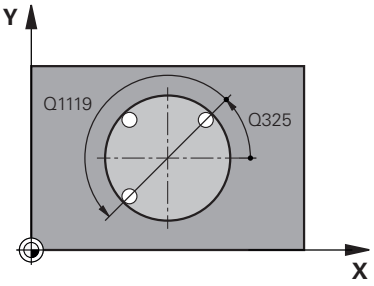
Cycle parameters

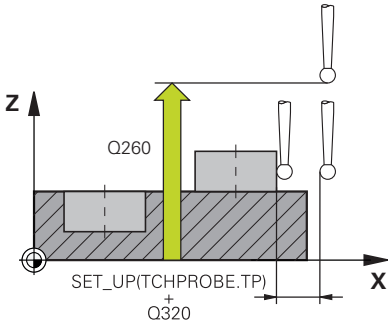


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal position of the first touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1116 Diameter of 1st position?**: Diameter of the first hole or stud. Input range: 0 to 9999.9999
- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal position of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position of the second touch point in the tool axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1117 Diameter of 2nd position?**: Diameter of the second hole or stud. Input range: 0 to 9999.9999
- **Q1115 Geometry type (0-3)?**: Define the geometry of the objects
  0: Position 1 = hole and position 2 = hole
  1: Position 1 = stud and position 2 = stud
  2: Position 1 = hole and position 2 = stud
  3: Position 1 = stud and position 2 = hole
- **Q423 Number of probes?** (absolute): Number of touch points on the diameter. Input range: 3 to 8
- **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: –360.000 to 360.000









- **Q1119 Arc angular length?**: Angular range in which the touch points are distributed. Input range: –359.999 to +360.000
- **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the touch points:
  –1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each object
  2: Move to clearance height before and after moving to each touch point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the program run.
  In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.
- **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
  0: Maintain the current tilting axes position
  1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the ball tip (TURN)
- **Q1120 Transfer position?**: Define which touch point will be used to correct the current preset:
  0: No correction
  1: Correction based on the 1st touch point
  2: Correction based on the 2nd touch point
  3: Correction based on the averaged touch point position

Example

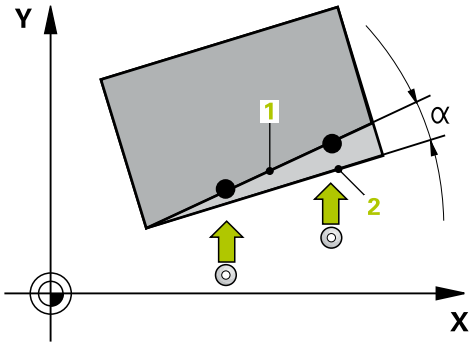
```
5 TCH PROBE 1410  PROBING TWO CIRCLES
   Q1100=+0   ;1ST POINT REF AXIS
   Q1101=+0   ;1ST POINT MINOR AXIS
   Q1102=+0   ;1ST POINT TOOL AXIS
   Q1116=0    ;DIAMETER 1
   Q1103=+0   ;2ND POINT REF AXIS
   Q1104=+0   ;2ND POINT MINOR AXIS
   Q1105=+0   ;2ND POINT TOOL AXIS
   Q1117=+0   ;DIAMETER 2
   Q1115=0    ;GEOMETRY TYPE
   Q423=4     ;NO. OF PROBE POINTS
   Q325=+0    ;STARTING ANGLE
   Q1119=+360 ;ANGULAR LENGTH
   Q320=+0    ;SET-UP CLEARANCE
   Q260=+100  ;CLEARANCE HEIGHT
   Q1125=+2   ;CLEAR. HEIGHT MODE
   Q309=+0    ;ERROR REACTION
   Q1126=+0   ;ALIGN ROTARY AXIS
   Q1120=+0   ;TRANSER POSITION
   Q1121=+0   ;CONFIRM ROTATION
```

- **Q1121 CONFIRM ROTATION?**: Define whether the control is to transfer the determined inclination as the basic rotation:
  0: No basic rotation
  1: Set basic rotation: The control saves the basic rotation
  2: Perform rotation of the rotary table: The control will make an entry in the corresponding **Offset** column of the preset table

17.6 Touch probe cycles 4xx: fundamentals

Characteristics common to all touch probe cycles for measuring workpiece misalignment

For Cycles 400, 401 and 402 you can define through parameter **Q307 Preset value for rotation angle** 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**.

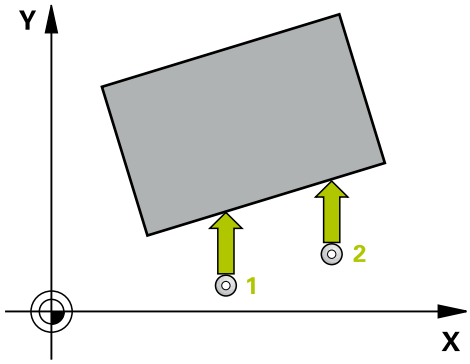


> These cycles do not work with 3-D Rot! In such a case, use Cycles 14xx. **Further information:** "Touch probe cycles 14xx: fundamentals", Page 601

17.7 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 line. With the basic rotation function, the control compensates the measured value.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 The touch probe then moves to the next touch point **2** and probes again.

4 The control returns the touch probe to the clearance height and performs the basic rotation it determined.

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

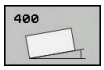
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

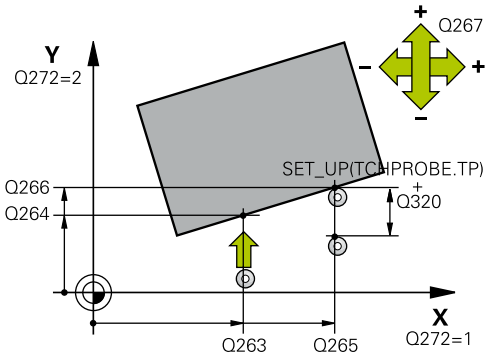
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

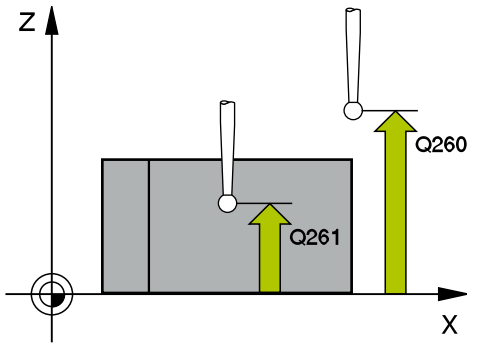
The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

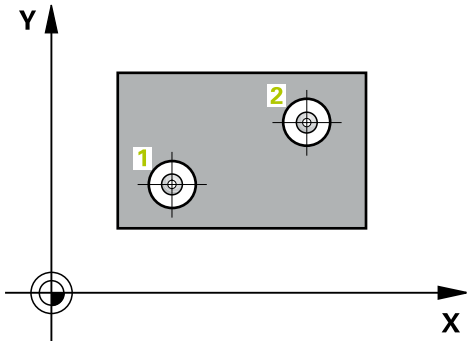
```
5 TCH PROBE 400  BASIC ROTATION
   Q263=+10   ;1ST POINT 1ST AXIS
   Q264=+3,5  ;1ST POINT 2ND AXIS
   Q265=+25   ;2ND POINT 1ST AXIS
   Q266=+2    ;2ND PNT IN 2ND AXIS
   Q272=+2    ;MEASURING AXIS
   Q267=+1    ;TRAVERSE DIRECTION
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE
   Q307=0     ;PRESET ROTATION ANG.
   Q305=0     ;NUMBER IN TABLE
```

- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: –360.000 to 360.000
- **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. If you enter **Q305**=0, the control automatically stores the determined basic rotation in the ROT menu of the Manual Operation mode. Input range: 0 to 99999

17.8 ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401)

Cycle run

Touch probe cycle 401 measures the center points of two holes. The control then calculates the angle between the reference axis of the working plane and the line connecting the hole center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the programmed center point of the first hole **1**.

2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.

3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.

4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.

5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

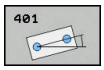
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control will reset an active basic rotation at the beginning of the cycle.

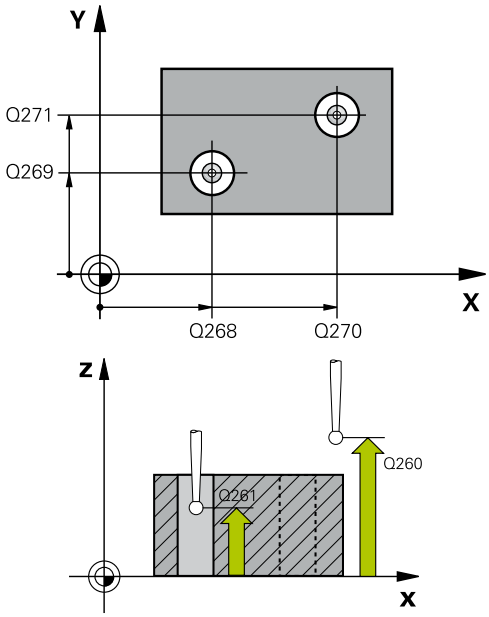
If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **Q268 1st hole: center in 1st axis?** (absolute): Center of the first hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q269 1st hole: center in 2nd axis?** (absolute): Center of the first hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q270 2nd hole: center in 1st axis?** (absolute): Center of the second hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q271 2nd hole: center in 2nd axis?** (absolute): Center of the second hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999



Example

```
5 TCH PROBE 401  ROT OF 2 HOLES
  Q268=-37   ;1ST CENTER 1ST AXIS
  Q269=+12   ;1ST CENTER 2ND AXIS
```

- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: –360.000 to 360.000
- **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row: Input range: 0 to 99 999
  Q305 = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.
  Q305 > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**).
  Q305 depends on the following parameters:
  Q337 = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, an entry is made in column **SPC**)
  Q337 = 0 and, at the same time, **Q402** = 1: Parameter **Q305** is not effective
  Q337 = 1: Parameter **Q305** is in effect as described above

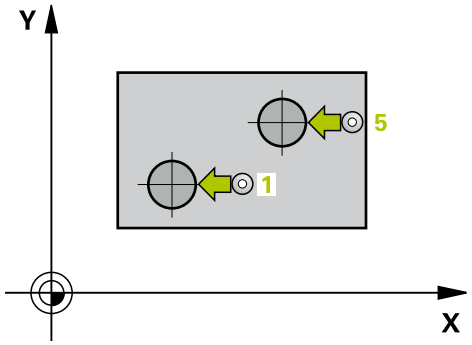
- **Q402 Basic rotation/alignment (0/1)**: Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of the rotary table:
  0: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**)
  1: Rotate the rotary table: An entry will be made in the **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column), in addition, the corresponding axis will be rotated
- **Q337 Set to zero after alignment?**: Define whether the control is to set the position display of the corresponding rotary axis to 0 after the alignment:
  0: The position display is not set to 0 after the alignment
  1: After the alignment, the position display is set to 0, provided you defined **Q402=1**

```
Q270=+75   ;2ND CENTER 1ST AXIS
Q271=+20   ;2ND CENTER 2ND AXIS
Q261=-5    ;MEASURING HEIGHT
Q260=+20   ;CLEARANCE HEIGHT
Q307=0     ;PRESET ROTATION ANG.
Q305=0     ;NUMBER IN TABLE
Q402=0     ;COMPENSATION
Q337=0     ;SET TO ZERO
```

17.9 ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402)

Cycle run

Touch probe cycle 402 measures the center points of two cylindrical studs. The control then calculates the angle between the reference axis of the working plane and the line connecting the stud center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



1 The control positions the touch probe at rapid traverse speed (value from FMAX column), using positioning logic (see "Executing touch probe cycles", Page 595), to the programmed center point of the first stud **1**.

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 control moves the touch probe to the entered **measuring height 2** and probes four points to determine the second stud center point.

5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

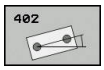
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control will reset an active basic rotation at the beginning of the cycle.

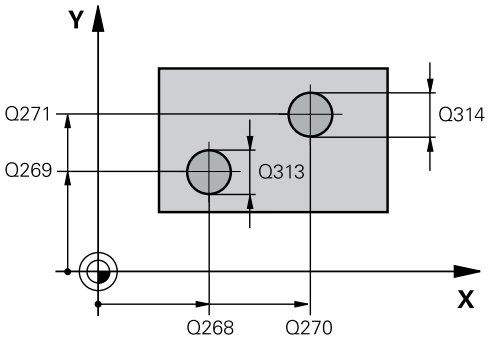
If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:

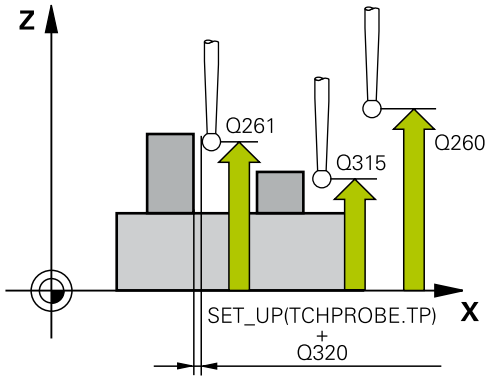
- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **Q268 1st stud: center in 1st axis?** (absolute): Center of the first stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q269 1st stud: center in 2nd axis?** (absolute): Center of the first stud in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q313 Diameter of stud 1?**: Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small. Input range: 0 to 99999.9999
- **Q261 Meas. height stud 1 in TS axis?** (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 is to be measured. Input range: –99999.9999 to 99999.9999
- **Q270 2nd stud: center in 1st axis?** (absolute): Center of the second stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q271 2nd stud: center in 2nd axis?** (absolute): Center of the second stud in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q313 Diameter of stud 2?**: Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small. Input range: 0 to 99999.9999
- **Q315 Meas. height stud 2 in TS axis?** (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 is to be measured. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

```
5 TCH PROBE 402  ROT OF 2 STUDS
  Q268=-37    ;1ST CENTER 1ST AXIS
  Q269=+12    ;1ST CENTER 2ND AXIS
  Q313=60     ;DIAMETER OF STUD 1
  Q261=-5     ;MEAS. HEIGHT STUD 1
  Q270=+75    ;2ND CENTER 1ST AXIS
  Q271=+20    ;2ND CENTER 2ND AXIS
  Q314=60     ;DIAMETER OF STUD 2
  Q315=-5     ;MEAS. HEIGHT STUD 2
  Q320=0      ;SET-UP CLEARANCE
  Q260=+20    ;CLEARANCE HEIGHT
  Q301=0      ;MOVE TO CLEARANCE
  Q307=0      ;PRESET ROTATION ANG.
  Q305=0      ;NUMBER IN TABLE
  Q402=0      ;COMPENSATION
  Q337=0      ;SET TO ZERO
```

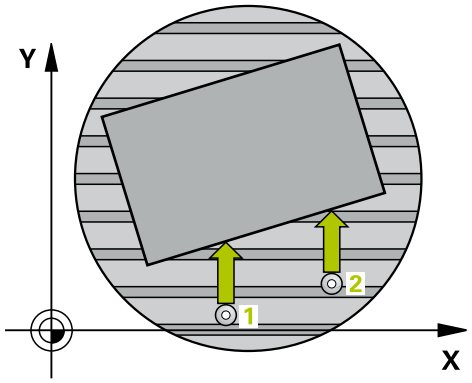
- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: –360.000 to 360.000
- **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row: Input range: 0 to 99 999
  Q305 = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.
  Q305 > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**).
  Q305 depends on the following parameters:
  Q337 = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, an entry is made in column **SPC**)
  Q337 = 0 and, at the same time, **Q402** = 1: Parameter **Q305** is not effective
  Q337 = 1: Parameter **Q305** is in effect as described above

- **Q402 Basic rotation/alignment (0/1)**: Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of the rotary table:
  0: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**)
  1: Rotate the rotary table: An entry will be made in the **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column), in addition, the corresponding axis will be rotated
- **Q337 Set to zero after alignment?**: Define whether the control is to set the position display of the corresponding rotary axis to 0 after the alignment:
  0: The position display is not set to 0 after the alignment
  1: After the alignment, the position display is set to 0, provided you defined **Q402=1**

17.10 ROT. IN 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 control 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 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 The touch probe then moves to the next touch point **2** and probes again.

4 The control 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 control is to set the determined rotation angle to 0 in the preset table or in the datum table.

Please note while programming:

NOTICE

Danger of collision!

If the control positions the rotary axis automatically, a collision might occur.

- Check for possible collisions between the tool and any elements positioned on the table
- Select the clearance height to prevent collisions

NOTICE

Danger of collision!

If you set parameter **Q312** Axis for compensating movement? to 0, then the cycle will automatically determine the rotary axis to be aligned (recommended setting). When doing so, it determines an angle that depends on the sequence of the touch points. The measured angle goes from the first to the second touch point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the probing points. The calculated angle lies in the range from –90° to +90°.

- After alignment, check the position of the rotary axis.

NOTICE

Danger of collision!

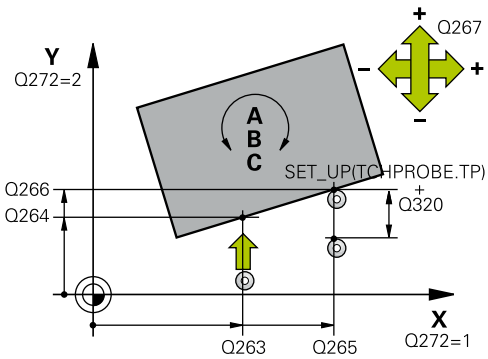
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

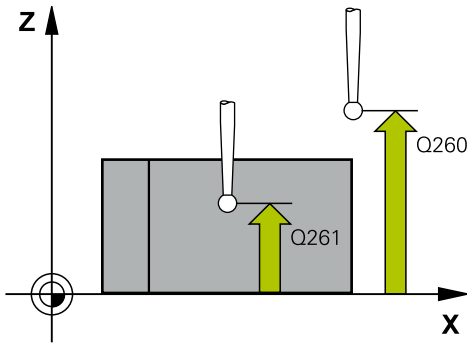
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

5 TCH PROBE 403 ROT IN ROTARY AXIS	
Q263=+0	;1ST POINT 1ST AXIS
Q264=+0	;1ST POINT 2ND AXIS
Q265=+20	;2ND PNT IN 1ST AXIS
Q266=+30	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q312=0	;COMPENSATION AXIS
Q337=0	;SET TO ZERO
Q305=1	;NUMBER IN TABLE
Q303=+1	;MEAS. VALUE TRANSFER
Q380=+90	;REFERENCE ANGLE

- **Q312 Axis for compensating movement?**: Specify the rotary axis in which the control is to compensate the measured misalignment:
  0: Automatic mode—the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting!
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C
- **Q337 Set to zero after alignment?**: Define whether the control should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment.
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment
- **Q305 Number in table?** Specify the number of the row in the preset table in which the control is to enter the basic rotation. Input range: 0 to 99999
  Q305 = 0: The rotary axis is zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.
  Q305 > 0: Specify the number of the row in the preset table in which the control is to zero the rotary axis. The control will make an entry in the **OFFSET** column of the preset table.
  Q305 depends on the following parameters:
  Q337 = 0: Parameter **Q305** is not effective
  Q337 = 1: Parameter **Q305** is effective as described above
  Q312 = 0: Parameter **Q305** is effective as described above
  Q312 > 0: The entry in **Q305** is ignored. The control will make an entry in the **OFFSET** column, in the row of the preset table that was active when the cycle was called.

- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
  0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q380 Ref. angle in ref. axis?**: Angle to which the control is to align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (**Q312** = 0 or 6). Input range: 0 to 360.000

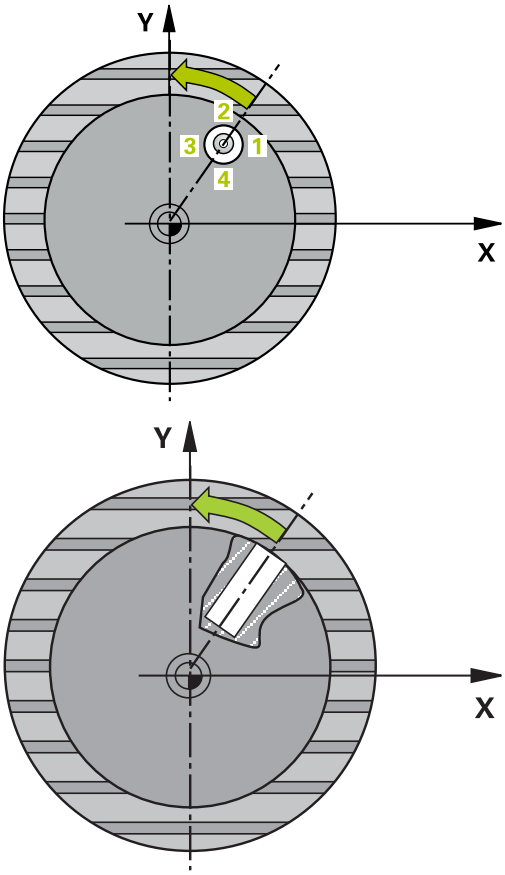
17.11 ROT. IN C AXIS (Cycle 405, DIN/ISO: G405

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 line of a hole
- the angular offset between the nominal position and the actual position of a hole center point

The control 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 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control 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 touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the hole center measured.

5 Finally the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center after compensation lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in parameter **Q150**.

Please note while programming:

NOTICE

Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control 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.

- Make sure that there is no material in the pocket/hole
- To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

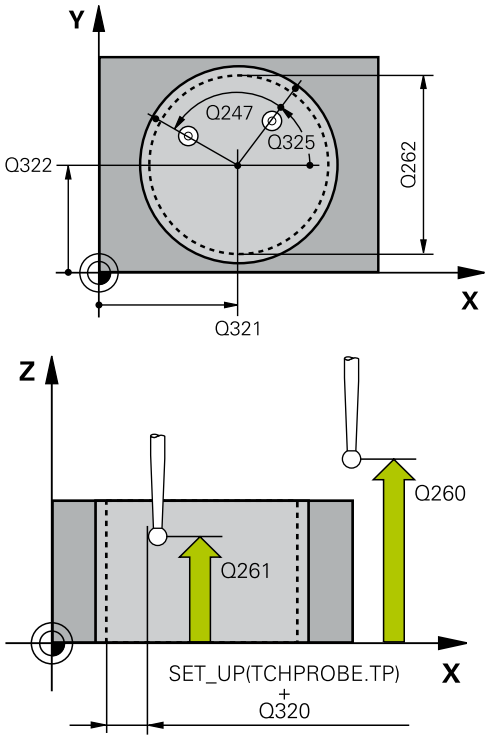
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The smaller the stepping angle, the less accurately the control can calculate the circle center point. Minimum input value: 5°.

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the hole in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point with the positive Y axis. If you program **Q322** not equal to zero 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center). Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: –360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999



Example

```
5 TCH PROBE 405  ROT IN C AXIS
   Q321=+50   ;CENTER IN 1ST AXIS
   Q322=+50   ;CENTER IN 2ND AXIS
   Q262=10    ;NOMINAL DIAMETER
   Q325=+0    ;STARTING ANGLE
   Q247=90    ;STEPPING ANGLE
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE
   Q337=0     ;SET TO ZERO
```

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q337 Set to zero after alignment?**:
  0: Set the display of the C axis to 0 and write value to **C_Offset** of the active row of the datum table
  >0: Write the measured angular offset to the datum table. Row number = value in **Q337**. If a C-axis shift is registered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative.

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

Cycle run

With touch probe cycle 404, you can set any basic rotation automatically during program run or save it to the preset table. You can also use Cycle 404 if you want to reset an active basic rotation.

NOTICE

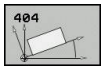
Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

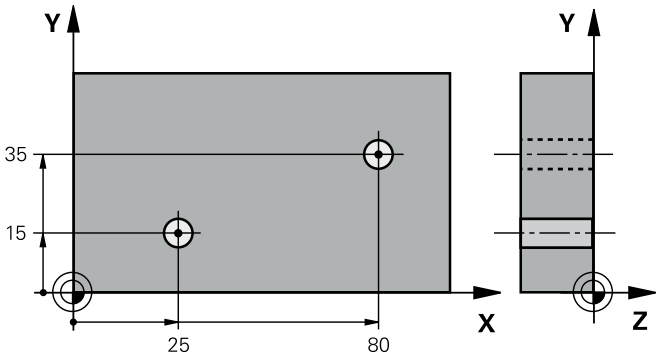


- **Q307 Preset value for rotation angle**: Angular value to which the basic rotation is to be set. Input range: –360.000 to 360.000
- **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. Input range: –1 to 99999. If you enter **Q305**=0 or **Q305**=–1, the control 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 row 0, write basic rotation to preset row 0 and activate preset 0
  >1 = Save the basic rotation to the specified preset. The preset is not activated.

Example

```
5 TCH PROBE 404  SET BASIC ROTATION
   Q307=+0       ;PRESET ROTATION ANG.
   Q305=-1       ;NUMBER IN TABLE
```

17.13 Example: Determining a basic rotation from two holes



0 BEGIN P GM CYC401 MM	
1 TOOL CALL 69 Z	
2 TCH PROBE 401 ROT OF 2 HOLES	
Q268=+25 ;1ST CENTER 1ST AXIS	Center of the 1st hole: X coordinate
Q269=+15 ;1ST CENTER 2ND AXIS	Center of the 1st hole: Y coordinate
Q270=+80 ;2ND CENTER 1ST AXIS	Center of the 2nd hole: X coordinate
Q271=+35 ;2ND CENTER 2ND AXIS	Center of the 2nd hole: Y coordinate
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+20 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q307=+0 ;PRESET ROTATION ANG.	Angle of the reference line
Q305=0 ;NUMBER IN TABLE	
Q402=1 ;COMPENSATION	Compensate misalignment by rotating the rotary table
Q337=1 ;SET TO ZERO	Set the display to zero after the alignment
3 CALL PGM 35K47	Call part program
4 END PGM CYC401 MM	

18

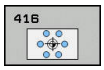
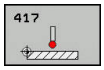
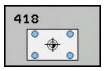
Touch Probe Cycles: Automatic Presetting

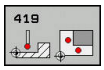
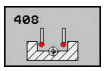
18.1 Fundamentals

Overview

The control offers twelve cycles for automatically finding presets and handling them as follows:

- Setting the determined values directly as display values
- Writing the determined values to the preset table
- Writing the determined values to a datum table

Soft key	Cycle	Page
	410 DATUM INSIDE RECTANGLE Measuring the inside length and width of a rectangle, and defining the center as preset	651
	411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as preset	655
	412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as preset	659
	413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as preset	664
	414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as preset	669
	415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as preset	674
	416 DATUM CIRCLE CENTER (2nd soft-key level) Measuring any three holes on a bolt hole circle, and defining the bolt-hole center as preset	679
	417 PRESET IN TS AXIS (2nd soft-key row) Measuring any position in the touch probe axis and defining it as preset	683
	418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as preset	685

Soft key	Cycle	Page
	419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as preset	690
	408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as preset	693
	409 RIDGE CENTER PRESET Measuring the outside width of a ridge, and define the ridge center point as the preset	697

The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Depending on the setting of the optional **CfgPresetSettings** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axis matches the tilting angles **3-D ROTATION**. If that is not the case, the control displays an error message.

Characteristics common to all touch probe cycles for presetting

You can also run the Touch Probe Cycles 408 to 419 during an active rotation (basic rotation or Cycle 10).

Preset and touch probe axis

The control determines the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Set preset in
Z	X and Y
Y	Z and X
X	Y and Z

Saving the calculated preset

In all cycles for presetting, you can use input parameters **Q303** and **Q305** to define how the control is to save the calculated preset:

- **Q305 = 0, Q303 = 1:**
  The control copies the active preset to row 0 and activates row 0, deleting simple transformations.
- **Q305 not equal to 0, Q303 = 0:**
  The result is written to the datum table, row **Q305**; **activate the datum with Cycle 7 in the NC program**
- **Q305 not equal to 0, Q303 = 1:**
  The result is written to the preset table, row **Q305**, the machine coordinate system is the reference system (REF coordinates); **activate the preset using Cycle 247 in the NC program**
- **Q305 not equal to 0, Q303 = -1**

This combination can only occur if you

- read in NC programs containing Cycles 410 to 418 created on a TNC 4xx
- read in NC 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 control 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**.

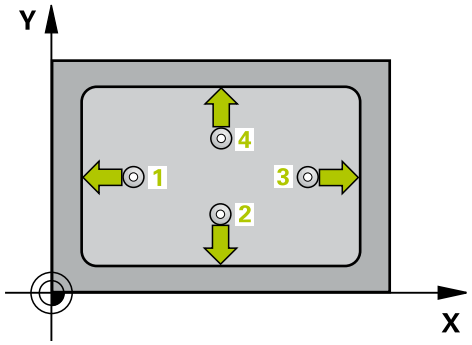
Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. You can use these parameters in your NC program. Note the table of result parameters listed with every cycle description.

18.2 PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410)

Cycle run

Touch probe cycle 410 finds the center of a rectangular pocket and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650)

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

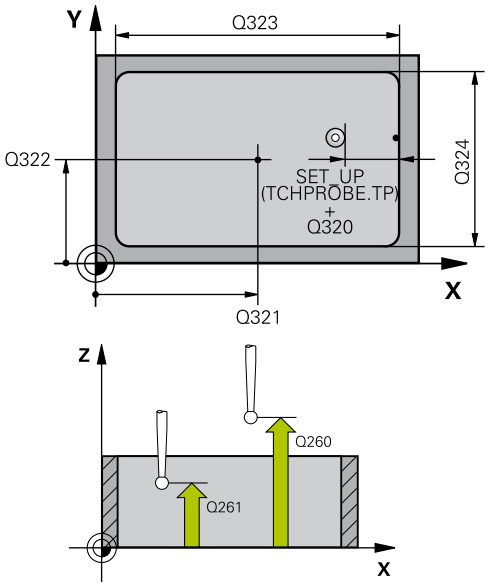
To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides. If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the pocket in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q323 First side length?** (incremental): Pocket length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q324 Second side length?** (incremental): Pocket length, parallel to the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999



Example

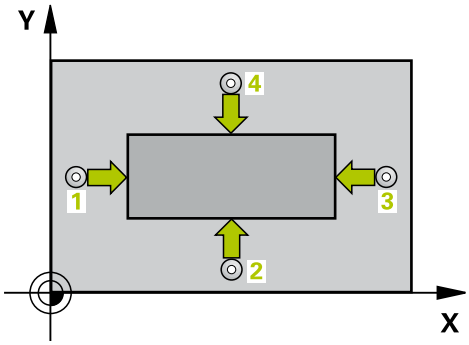
```
5 TCH PROBE 410  PRESET INSIDE
  RECTAN
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q323=60    ;FIRST SIDE LENGTH
  Q324=20    ;2ND SIDE LENGTH
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q305=10    ;NUMBER IN TABLE
  Q331=+0    ;PRESET
  Q332=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
  Q383=+50   ;2ND CO. FOR TS AXIS
  Q384=+0    ;3RD CO. FOR TS AXIS
  Q333=+1    ;PRESET
```

- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

18.3 PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411)

Cycle run

Touch probe cycle 411 finds the center of a rectangular stud and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650)

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

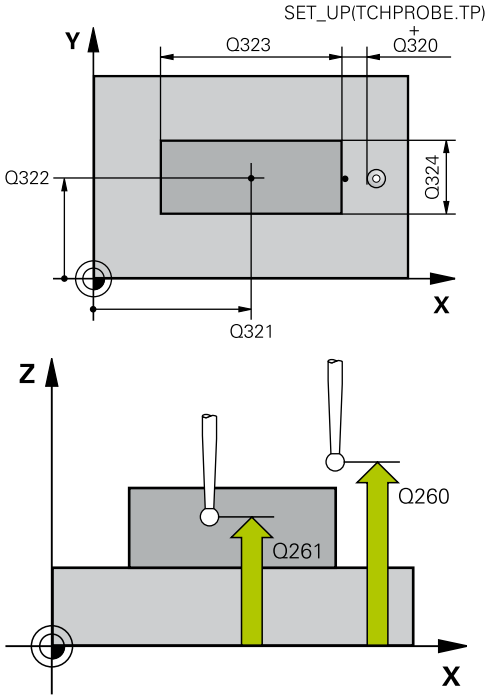
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q323 First side length?** (incremental): Stud length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q324 Second side length?** (incremental): Stud length, parallel to the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999



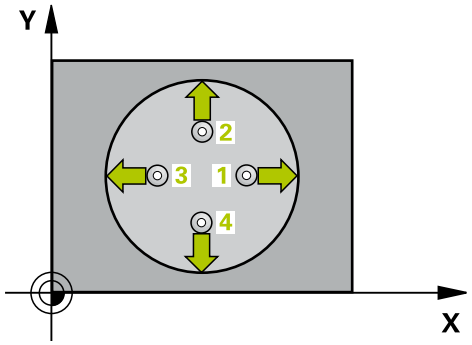
Example

```
5 TCH PROBE 411  PRESET OUTS.
  RECTAN
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q323=60    ;FIRST SIDE LENGTH
  Q324=20    ;2ND SIDE LENGTH
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q305=0     ;NUMBER IN TABLE
  Q331=+0    ;PRESET
  Q332=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
  Q383=+50   ;2ND CO. FOR TS AXIS
  Q384=+0    ;3RD CO. FOR TS AXIS
  Q333=+1    ;PRESET
```

- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

18.4 PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412)

Cycle run



Touch probe cycle 412 finds the center of a circular pocket (hole) and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control 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 touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the actual values in the Q parameters listed below

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

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 set-up clearance do not permit pre-positioning in the proximity of the touch points, the control 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.

- Positioning of the touch points
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

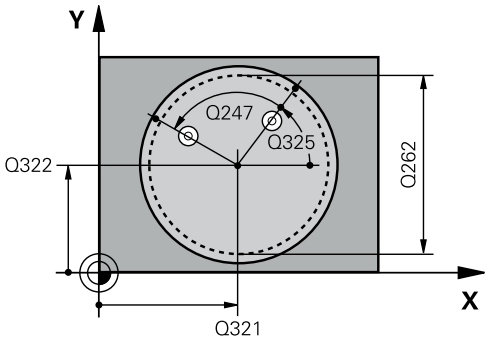
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

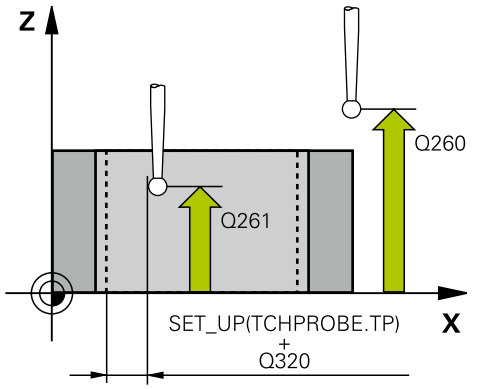
The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset.
Minimum input value: 5°

Program the stepping angle to be less than 90°, input range: –120° to 120°

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the pocket in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: –360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999





Example

```
5 TCH PROBE 412  PRESET INSIDE
  CIRCLE
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q262=75    ;NOMINAL DIAMETER
  Q325=+0    ;STARTING ANGLE
  Q247=+60   ;STEPPING ANGLE
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q305=12    ;NUMBER IN TABLE
```

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

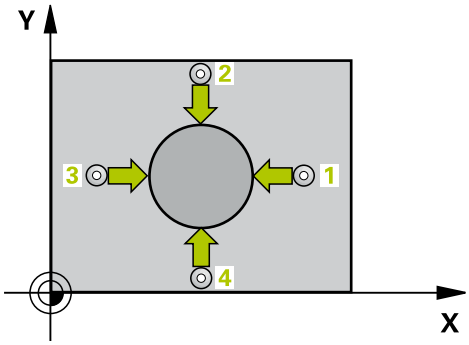
```
Q331=+0     ;PRESET
Q332=+0     ;PRESET
Q303=+1     ;MEAS. VALUE TRANSFER
Q381=1      ;PROBE IN TS AXIS
Q382=+85    ;1ST CO. FOR TS AXIS
Q383=+50    ;2ND CO. FOR TS AXIS
Q384=+0     ;3RD CO. FOR TS AXIS
Q333=+1     ;PRESET
Q423=4      ;NO. OF PROBE POINTS
Q365=1      ;TYPE OF TRAVERSE
```

- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301**=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

18.5 PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413)

Cycle run

Touch probe cycle 413 finds the center of a circular stud and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control 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 touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the actual values in the Q parameters listed below
6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

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.

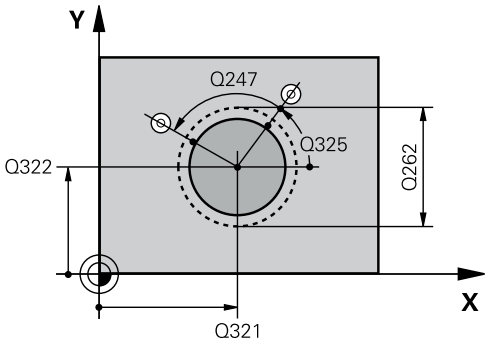
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

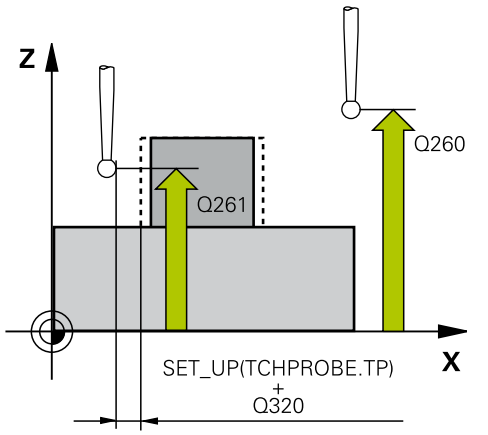
The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°, input range: –120° to 120°

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: –360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

```
5 TCH PROBE 413  PRESET OUTS.
  CIRCLE
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q262=75    ;NOMINAL DIAMETER
  Q325=+0    ;STARTING ANGLE
  Q247=+60   ;STEPPING ANGLE
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q305=15    ;NUMBER IN TABLE
  Q331=+0    ;PRESET
  Q332=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
  Q383=+50   ;2ND CO. FOR TS AXIS
  Q384=+0    ;3RD CO. FOR TS AXIS
  Q333=+1    ;PRESET
  Q423=4     ;NO. OF PROBE POINTS
  Q365=1     ;TYPE OF TRAVERSE
```

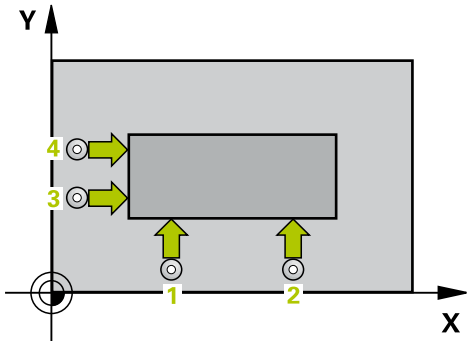
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301**=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

18.6 PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414)

Cycle run

Touch probe cycle 414 finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1** (see figure at right). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.

3 The touch probe then moves to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the coordinates of the determined corner in the Q parameters listed below

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

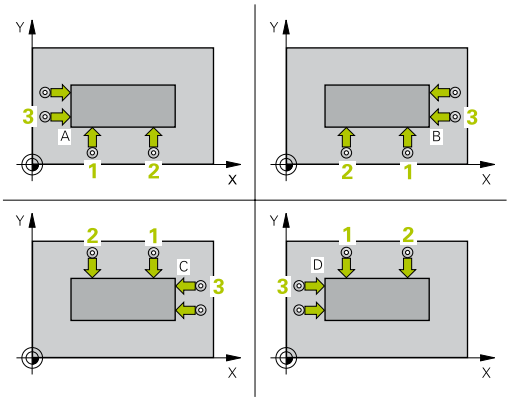
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control always measures the first line in the direction of the minor axis of the working plane.

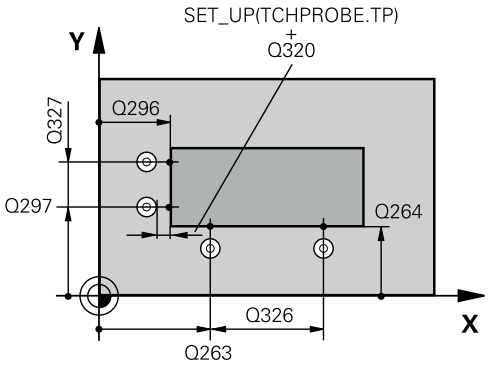
By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see figure at the right and table below).

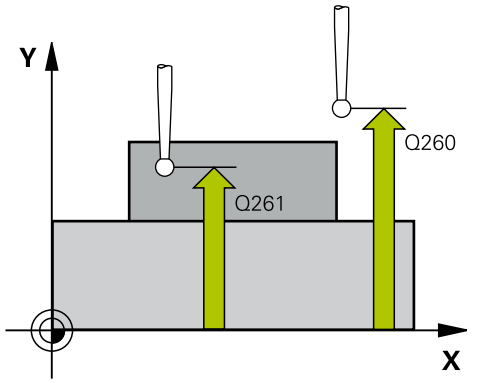


Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

```
5 TCH PROBE 414  PRESET INSIDE CORNER
   Q263=+37   ;1ST POINT 1ST AXIS
   Q264=+7    ;1ST POINT 2ND AXIS
   Q326=50    ;SPACING IN 1ST AXIS
   Q296=+95   ;3RD PNT IN 1ST AXIS
   Q297=+25   ;3RD PNT IN 2ND AXIS
   Q327=45    ;SPACING IN 2ND AXIS
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE
   Q304=0     ;BASIC ROTATION
   Q305=7     ;NUMBER IN TABLE
   Q331=+0    ;PRESET
```

- **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the corner coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the corner. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the corner. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

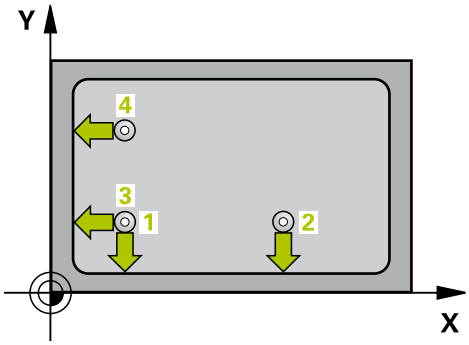
```
Q332=+0     ;PRESET
Q303=+1     ;MEAS. VALUE TRANSFER
Q381=1      ;PROBE IN TS AXIS
Q382=+85    ;1ST CO. FOR TS AXIS
Q383=+50    ;2ND CO. FOR TS AXIS
Q384=+0     ;3RD CO. FOR TS AXIS
Q333=+1     ;PRESET
```

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

18.7 PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415)

Cycle run

Touch probe cycle 415 finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1** (see figure at right). The control offsets the touch probe in the principal and secondary axes by the set-up clearance **Q320** + **SET_UP** + ball-tip radius (in the direction opposite the respective traverse direction)

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The probing direction is derived from the number by which you identify the corner.

3 The touch probe moves to the next touch point **2**; the control offsets the touch probe in the secondary axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the second probing operation

4 The control positions the touch probe at touch point **3** (same positioning logic as for the first touch point) and performs the probing operation there

5 Then the touch probe moves to touch point **4**. The control offsets the touch probe in the principal axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the fourth probing operation

6 Finally, the control retracts the touch probe to the clearance height. It processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the coordinates of the determined corner in the Q parameters listed below

7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

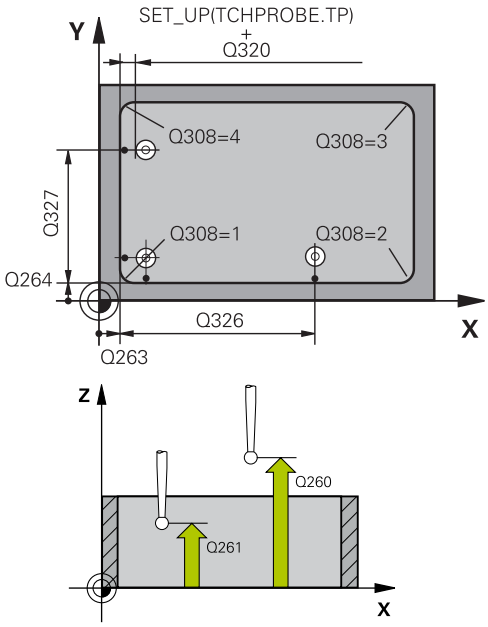
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control always measures the first line in the direction of the minor axis of the working plane.

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the corner in the principal axis of the working plane Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the corner in the secondary axis of the working plane Input range: –99999.9999 to 99999.9999
- **Q326 Spacing in 1st axis?** (incremental): Distance between the corner and second measuring point in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q327 Spacing in 2nd axis?** (incremental): Distance between the corner and fourth measuring point in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q308 Corner? (1/2/3/4)**: Number identifying the corner which the control is to set as the preset. Input range: 1 to 4
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation



Example

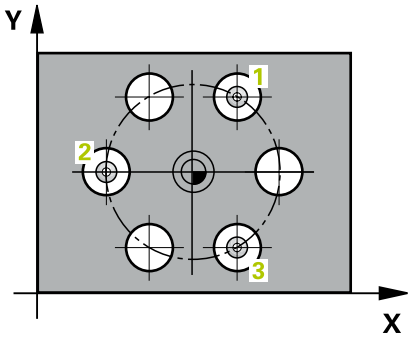
```
5 TCH PROBE 415  PRESET OUTS. CORNER
  Q263=+37   ;1ST POINT 1ST AXIS
  Q264=+7    ;1ST POINT 2ND AXIS
  Q326=50    ;SPACING IN 1ST AXIS
  Q327=45    ;SPACING IN 2ND AXIS
  Q308=+1    ;CORNER
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q304=0     ;BASIC ROTATION
  Q305=7     ;NUMBER IN TABLE
  Q331=+0    ;PRESET
  Q332=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
  Q383=+50   ;2ND CO. FOR TS AXIS
  Q384=+0    ;3RD CO. FOR TS AXIS
  Q333=+1    ;PRESET
```

- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the corner coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the corner. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the corner. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

18.8 PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416)

Cycle run



Touch probe cycle 416 finds the center of a bolt hole circle by measuring three holes, and defines the determined center as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the programmed center point of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the actual values in the Q parameters listed below
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Please note while programming:

NOTICE

Danger of collision!

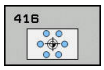
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

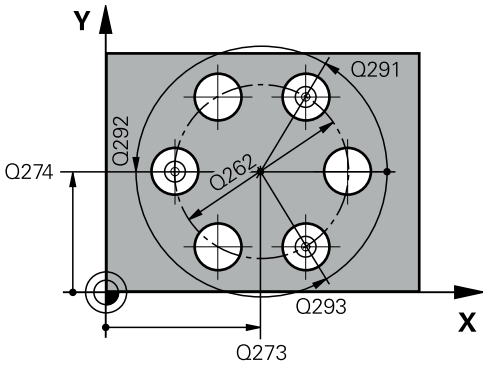
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

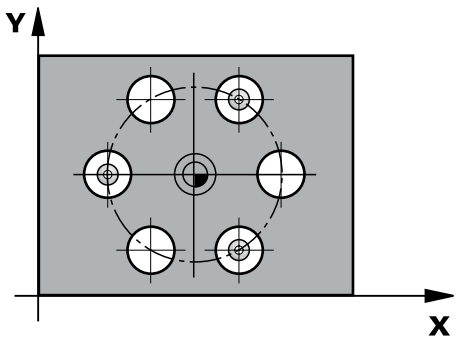
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input range: –0 to 99999.9999
- **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range: –360.0000 to 360.0000
- **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range: –360.0000 to 360.0000
- **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range: –360.0000 to 360.0000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: –99999.9999 to 99999.9999





Example

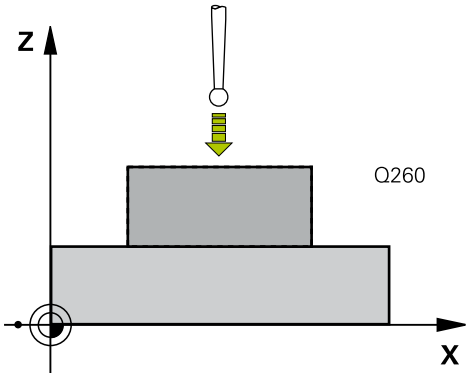
```
5 TCH PROBE 416  PRESET CIRCLE
  CENTER
  Q273=+50   ;CENTER IN 1ST AXIS
  Q274=+50   ;CENTER IN 2ND AXIS
  Q262=90    ;NOMINAL DIAMETER
  Q291=+34   ;ANGLE OF 1ST HOLE
  Q292=+70   ;ANGLE OF 2ND HOLE
  Q293=+210  ;ANGLE OF 3RD HOLE
  Q261=-5    ;MEASURING HEIGHT
  Q260=+20   ;CLEARANCE HEIGHT
  Q305=12    ;NUMBER IN TABLE
  Q331=+0    ;PRESET
  Q332=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
  Q383=+50   ;2ND CO. FOR TS AXIS
  Q384=+0    ;3RD CO. FOR TS AXIS
  Q333=+1    ;PRESET
  Q320=0     ;SET-UP CLEARANCE
```

- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999

18.9 PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417)

Cycle run

Touch probe cycle 417 measures any coordinate in the touch probe axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at the programmed touch point **1**. The control offsets the touch probe by the set-up 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 touch point **1** and measures the actual position with a simple probing movement

3 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the actual value in the Q parameter listed below

Parameter number	Meaning
Q160	Actual value of measured point

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

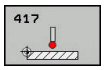
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

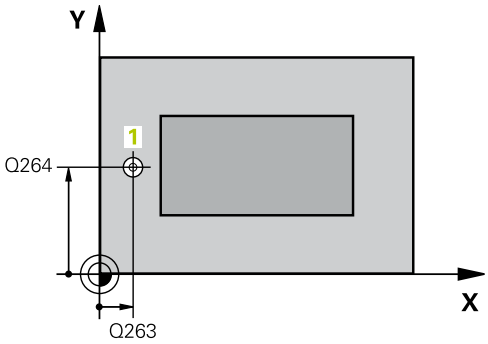
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

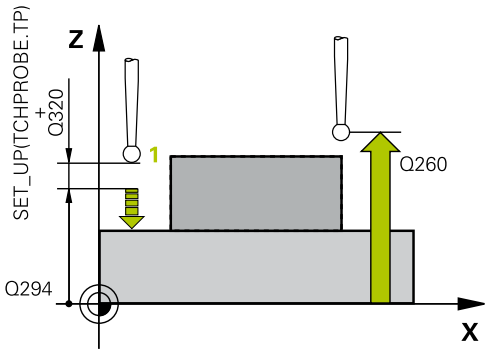
The control then sets the preset in this axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).





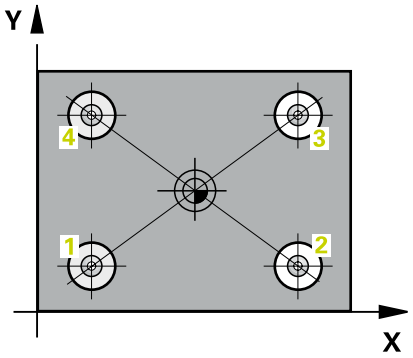
Example

```
5 TCH PROBE 417  PRESET IN TS AXIS
  Q263=+25   ;1ST POINT 1ST AXIS
  Q264=+25   ;1ST POINT 2ND AXIS
  Q294=+25   ;1ST POINT 3RD AXIS
  Q320=0     ;SET-UP CLEARANCE
  Q260=+50   ;CLEARANCE HEIGHT
  Q305=0     ;NUMBER IN TABLE
  Q333=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
```

18.10 PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418)

Cycle run

Touch probe cycle 418 calculates the intersection of the lines connecting two opposite hole center points and sets the preset at the point of intersection. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the center point of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The control repeats this step for holes **3** and **4**.
6 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650). The control calculates the preset 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 control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

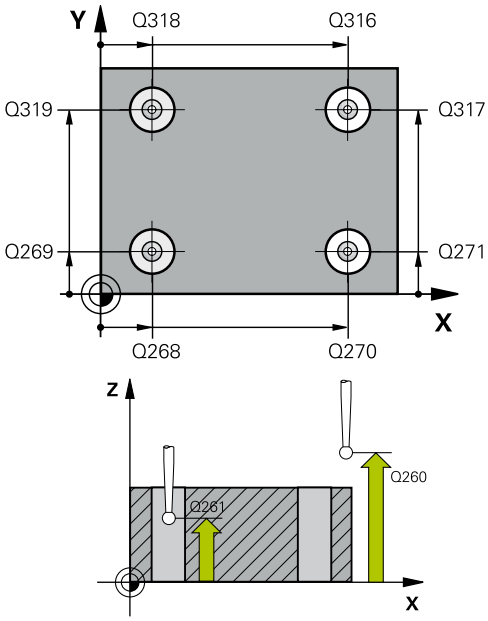
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

- **Q268 1st hole: center in 1st axis?** (absolute): Center of the first hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q269 1st hole: center in 2nd axis?** (absolute): Center of the first hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q270 2nd hole: center in 1st axis?** (absolute): Center of the second hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q271 2nd hole: center in 2nd axis?** (absolute): Center of the second hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q316 3rd hole: Center in 1st axis?** (absolute): Center of the third hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q317 3rd hole: Center in 2nd axis?** (absolute): Center of the third hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q318 4th hole: Center in 1st axis?** (absolute): Center of the fourth hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q319 4th hole: Center in 2nd axis?** (absolute): Center of the fourth hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999



Example

```
5 TCH PROBE 418  PRESET FROM 4 HOLES
   Q268=+20   ;1ST CENTER 1ST AXIS
   Q269=+25   ;1ST CENTER 2ND AXIS
   Q270=+150  ;2ND CENTER 1ST AXIS
   Q271=+25   ;2ND CENTER 2ND AXIS
   Q316=+150  ;3RD CENTER 1ST AXIS
   Q317=+85   ;3RD CENTER 2ND AXIS
   Q318=+22   ;4TH CENTER 1ST AXIS
   Q319=+80   ;4TH CENTER 2ND AXIS
   Q261=-5    ;MEASURING HEIGHT
   Q260=+10   ;CLEARANCE HEIGHT
   Q305=12    ;NUMBER IN TABLE
   Q331=+0    ;PRESET
   Q332=+0    ;PRESET
   Q303=+1    ;MEAS. VALUE TRANSFER
   Q381=1     ;PROBE IN TS AXIS
   Q382=+85   ;1ST CO. FOR TS AXIS
   Q383=+50   ;2ND CO. FOR TS AXIS
   Q384=+0    ;3RD CO. FOR TS AXIS
   Q333=+0    ;PRESET
```

- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the point of intersection of the connecting lines; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the intersection of the connecting lines. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should set the intersection of the connecting lines. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

18.11 PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419)

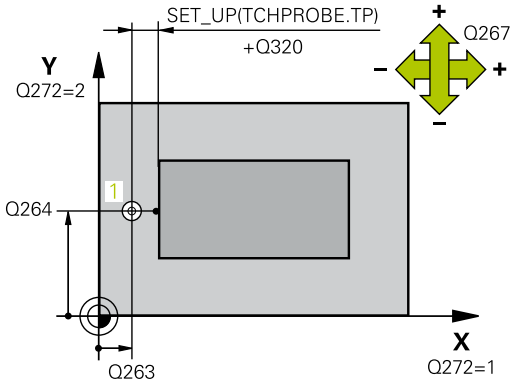
Cycle run

Touch probe cycle 419 measures any coordinate in the a selectable axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at the programmed touch point 1. The control offsets the touch probe by the safety clearance in the direction opposite to the programmed direction of probing

2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.

3 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650)



Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

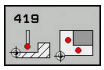
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

If you want to save the preset in 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.

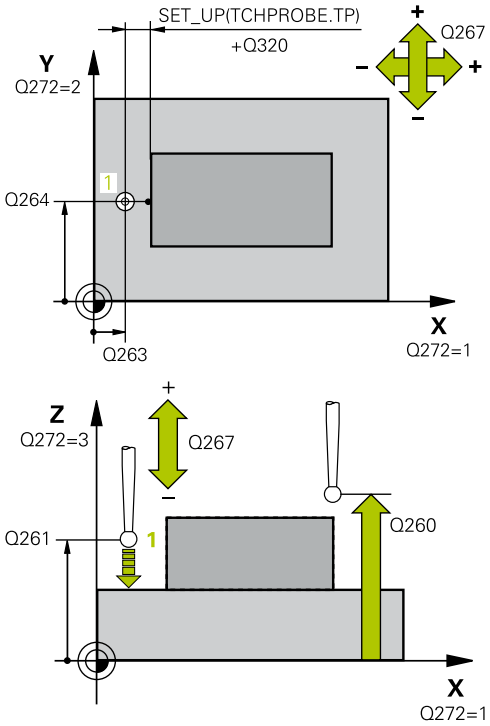
Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis

Axis assignment		
Active touch probe axis: Q272 = 3	**Associated reference axis: Q272 = 1**	**Associated minor axis: Q272 = 2**
Z	X	Y
Y	Z	X
X	Y	Z

- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction



Example

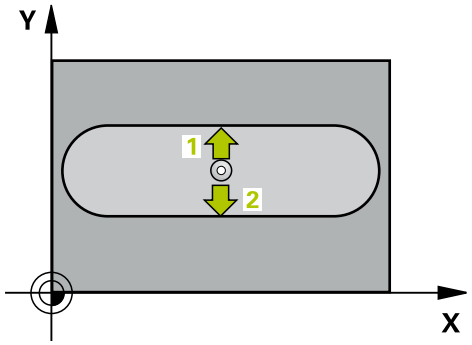
```
5 TCH PROBE 419  PRESET IN ONE AXIS
  Q263=+25   ;1ST POINT 1ST AXIS
  Q264=+25   ;1ST POINT 2ND AXIS
  Q261=+25   ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+50   ;CLEARANCE HEIGHT
  Q272=+1    ;MEASURING AXIS
  Q267=+1    ;TRAVERSE DIRECTION
  Q305=0     ;NUMBER IN TABLE
  Q333=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
```

- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q333 New preset?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  –1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 650)
  0: Write the determined preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

18.12 SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408)

Cycle run

Touch probe cycle 408 finds the center of a slot and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.

4 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the actual values in the Q parameters listed below

5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

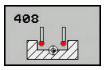
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width. If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control 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.

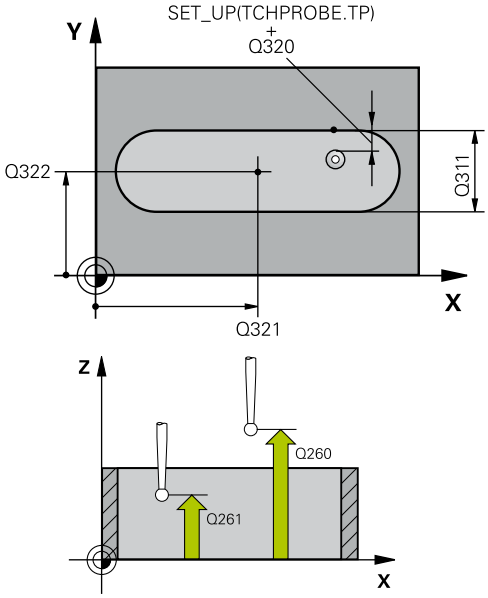
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the slot in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the slot in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q311 Width of slot?** (incremental): Width of the slot, regardless of its position in the working plane. Input range: 0 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated slot center. Default setting = 0. Input range: –99999.9999 to 99999.9999



Example

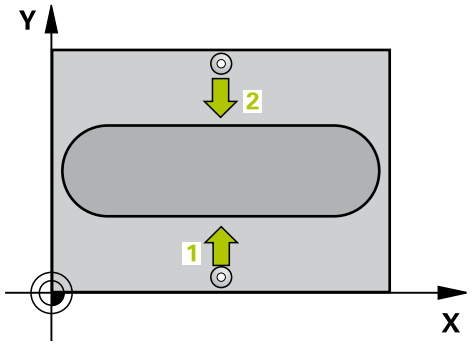
```
5 TCH PROBE 408  SLOT CENTER
  PRESET
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q311=25    ;SLOT WIDTH
  Q272=1     ;MEASURING AXIS
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q305=10    ;NUMBER IN TABLE
  Q405=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
  Q383=+50   ;2ND CO. FOR TS AXIS
  Q384=+0    ;3RD CO. FOR TS AXIS
  Q333=+1    ;PRESET
```

- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
  0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

18.13 RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409)

Cycle run

Touch probe cycle 409 finds the center of a ridge and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.

4 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 650) and saves the actual values in the Q parameters listed below

5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

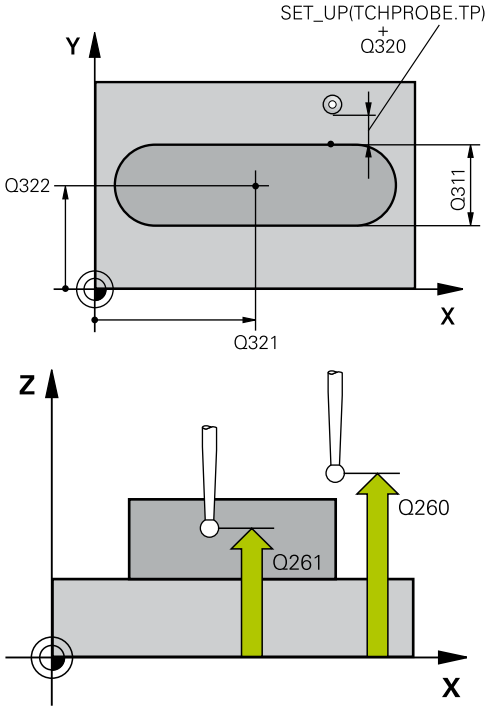
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the ridge in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the ridge in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q311 Ridge width?** (incremental): Width of the ridge, regardless of its position in the working plane. Input range: 0 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.



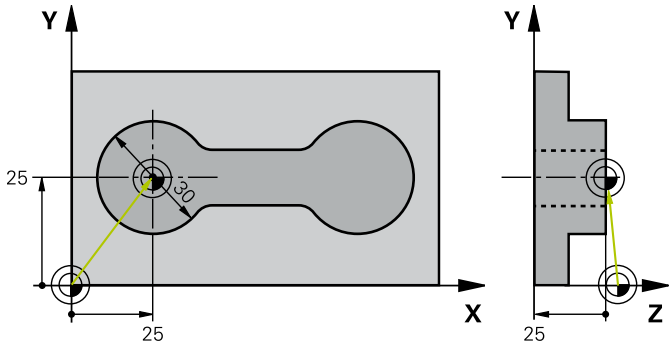
Example

```
5 TCH PROBE 409  RIDGE CENTER
  PRESET
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q311=25    ;RIDGE WIDTH
  Q272=1     ;MEASURING AXIS
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q305=10    ;NUMBER IN TABLE
  Q405=+0    ;PRESET
  Q303=+1    ;MEAS. VALUE TRANSFER
  Q381=1     ;PROBE IN TS AXIS
  Q382=+85   ;1ST CO. FOR TS AXIS
```

- **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated ridge center. Default setting = 0. Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
  0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1. Input range: –99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: –99999.9999 to 99999.9999

```
Q383=+50   ;2ND CO. FOR TS AXIS
Q384=+0    ;3RD CO. FOR TS AXIS
Q333=+1    ;PRESET
```

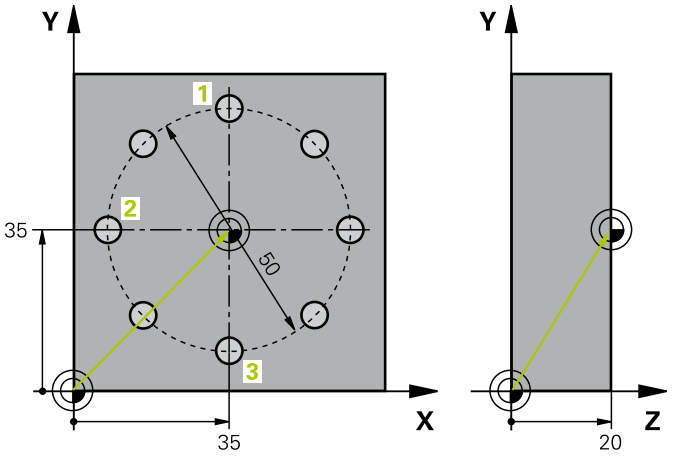
18.14 Example: Presetting in center of a circular segment and on top surface of workpiece



0 BEGIN PGM CYC413 MM		
1 TOOL CALL 69 Z		
2 TCH PROBE 413 PRESET OUTS. CIRCLE		
Q321=+25	;CENTER IN 1ST AXIS	Center of circle: X coordinate
Q322=+25	;CENTER IN 2ND AXIS	Center of circle: Y coordinate
Q262=30	;NOMINAL DIAMETER	Circle diameter
Q325=+90	;STARTING ANGLE	Polar coordinate angle for 1st touch point
Q247=+45	;STEPPING ANGLE	Stepping angle for calculating the starting points 2 to 4
Q261=-5	;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q320=2	;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+10	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q301=0	;MOVE TO CLEARANCE	Do not move to clearance height between measuring points
Q305=0	;NUMBER IN TABLE	Set display
Q331=+0	;PRESET	Set the display in X to 0
Q332=+10	;PRESET	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 preset 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	;PRESET	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		

18.15 Example: Presetting on top surface of workpiece and in center of a bolt hole circle

The control is to write the measured bolt-hole circle center to 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 POBE 417 PRESET IN TS AXIS	Cycle definition for presetting in the touch probe axis
Q263=+7.5 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=0 ;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+50 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NUMBER IN TABLE	Write Z coordinate in line 1
Q333=+0 ;PRESET	Set touch-probe axis to 0
Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated preset referenced to the machine-based coordinate system (REF system)
3 TCH PROBE 416 PRESET CIRCLE CENTER	
Q273=+35 ;CENTER IN 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35 ;CENTER IN 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50 ;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90 ;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center **1**
Q292=+180 ;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center **2**
Q293=+270 ;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center **3**
Q261=+15 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NUMBER IN TABLE	Write center of bolt hole circle (X and Y) to row 1
Q331=+0 ;PRESET	
Q332=+0 ;PRESET	

Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated preset referenced to the machine-based coordinate system (REF system)
Q381=0 ;PROBE IN TS AXIS	Do not set a preset in the touch probe axis
Q382=+0 ;1ST CO. FOR TS AXIS	No function
Q383=+0 ;2ND CO. FOR TS AXIS	No function
Q384=+0 ;3RD CO. FOR TS AXIS	No function
Q333=+0 ;PRESET	No function
Q320=0 ;SET-UP CLEARANCE.	Safety clearance in addition to SET_UP column
4 CYCL DEF 247 PRESETTING	Activate new preset with Cycle 247
Q339=1 ;PRESET NUMBER	
6 CALL PGM 35KLZ	Call part program
7 END PGM CYC416 MM	

19

Touch Probe Cycles: Automatic Workpiece Inspection

19.1 Fundamentals

Overview

NOTICE

Danger of collision!

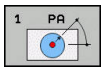
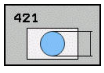
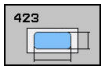
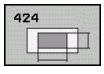
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

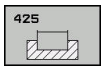
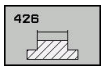
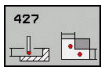
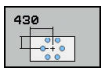
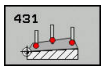
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The control offers twelve cycles for measuring workpieces automatically.

Soft key	Cycle	Page
	0 REFERENCE PLANE Measuring a coordinate in a selectable axis	712
	1 POLAR PRESET Measuring a point, probing direction defined by an angle	713
	420 MEASURE ANGLE Measuring an angle in the working plane	714
	421 MEASURE HOLE Measuring the position and diameter of a hole	717
	422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud	722
	423 MEASURE RECTANGLE INSIDE Measuring the position, length, and width of a rectangular pocket	727
	424 MEASURE RECTANGLE OUTSIDE Measuring the position, length, and width of a rectangular stud	731

Soft key	Cycle	Page
	425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width	734
	426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge	737
	427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis	740
	430 MEAS. BOLT HOLE (2nd soft-key row) Measuring the position and diameter of a bolt hole circle	743
	431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane	746

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 control record the measurement results in a log. In the respective probing cycle you can define if the control 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 control by default saves the data as an ASCII file. The control 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

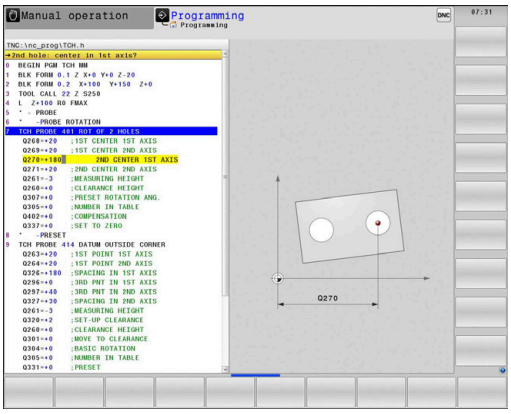
Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor 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 control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. Deviations from the nominal values are saved in parameters **Q161** to **Q166**. Note the table of result parameters listed with every cycle description.

During cycle definition, the control also shows the result parameters for the respective cycle in a help graphic (see figure at right). The highlighted result parameter belongs to that input parameter.



Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters **Q180** to **Q182**.

Class of results	Parameter value
Measurement results are within tolerance	**Q180** = 1
Rework is required	**Q181** = 1
Scrap	**Q182** = 1

The control sets the rework or scrap marker as soon as one of the measuring values is out of tolerance. To determine which of the measuring results is out of tolerance, check the measuring log, or compare the respective measuring results (**Q150** to **Q160**) with their limit values.

In Cycle 427 the control assumes by default 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 control also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

With most cycles for workpiece inspection, you can have the control 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 default value 0 for this parameter set this parameter unchanged.

Tool monitoring

With some cycles for workpiece inspection, you can have the control perform tool monitoring. The control then monitors whether

- the tool radius should be compensated due to 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 only works

- if the tool table is active.
- if tool monitoring is switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. To enter the tool name, press the associated soft key. The control no longer displays the right single quotation mark.

If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling tool: If you refer to a milling cutter in parameter **Q330**, the appropriate values are compensated in the following way: The control basically always compensates the tool radius in column DR of the tool table, even if the measured deviation is within the specified tolerance. You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Turning tool: (only applies to Cycles 421, 422, 427) If you reference a turning tool in parameter **Q330**, the appropriate values in row DZL and DXL, respectively, will be compensated. The control also 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: rework required).

If you want to compensate the values for an indexed tool with a tool name, program the following:

- **QS0** = "TOOL NAME"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool

Tool breakage monitoring

This function only works

- if the tool table is active.
- if tool monitoring is switched on in the cycle (set **Q330** unequal to 0)
- if the breakage tolerance RBREAK for the tool number entered in the table is greater than 0

Further information: User's Manual for Setup, Testing and Running NC Programs

The control will output an error message and stop the 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 control transfers all measurement results, which reference the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system, to the result parameters and the log file.

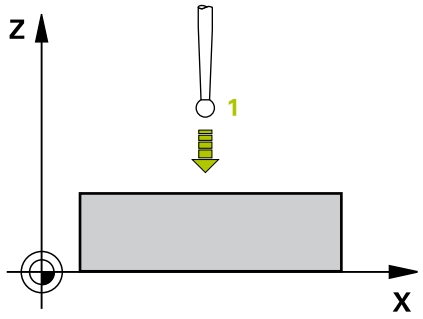
19.2 REFERENCE PLANE (Cycle 0, DIN/ISO: G55)

Cycle run

1 In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.

2 Next, the touch probe performs probing at the probing feed rate (**F** column). The probing direction must be defined in the cycle.

3 After the control has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. In addition, the control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**. For the values in these parameters the control does not account for the stylus length and radius.



Please note while programming:

NOTICE

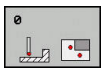
Danger of collision!

The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position in order to avoid a collision when the programmed pre-positioning point is approached

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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?**: Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction. Confirm with the **ENT** key. Input range: All NC axes
- **Position value?**: Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input range: –99999.9999 to 99999.9999
- To confirm your input, press the **ENT** key.

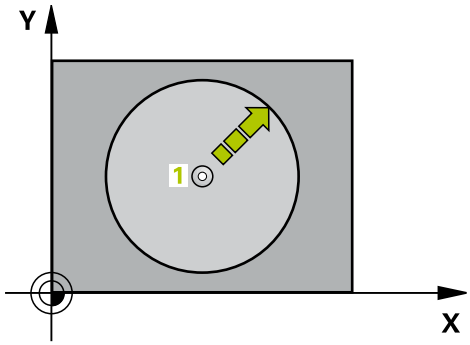
Example

```
67 TCH PROBE 0.0  REF. PLANE Q5 X-
68 TCH PROBE 0.1  X+5 Y+0 Z-5
```

19.3 POLAR PRESET (Cycle 1)

Cycle run

Touch probe cycle 1 measures any position on the workpiece in any probing direction.



1 In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.

2 Next, the touch probe performs probing at the probing feed rate (**F** column). During probing, the control moves the touch probe simultaneously in two axes (depending on the probing angle). Use polar angles to define the probing direction in the cycle.

3 After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**

Please note while programming:

NOTICE

Danger of collision!

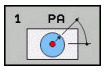
The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position in order to avoid a collision when the programmed pre-positioning point is approached

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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?**: Specify the probing axis with the axis key or the alphabetic keyboard. Confirm with the **ENT** key. Input range: **X**, **Y** or **Z**
- **Probing angle?**: Angle, measured from the probing axis, at which the touch probe is to move. Input range: -180.0000 to 180.0000
- **Position value?**: Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input range: –99999.9999 to 99999.9999
- To confirm your input, press the **ENT** key.

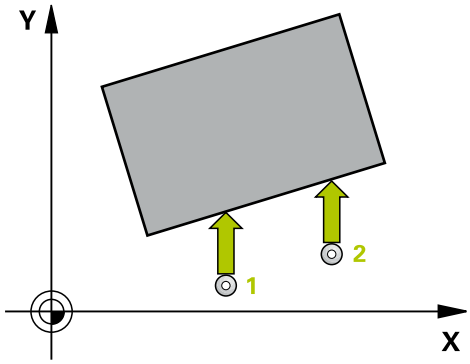
Example

```
67 TCH PROBE 1.0  POLAR PRESET
68 TCH PROBE 1.1  X ANGLE: +30
69 TCH PROBE 1.2  X+5 Y+0 Z-5
```

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

Cycle run

Touch Probe Cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at the programmed touch point 1. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 The touch probe then moves to the next touch point 2 and probes again.

4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

Parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the machining plane.

Please note while programming:

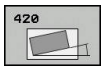
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

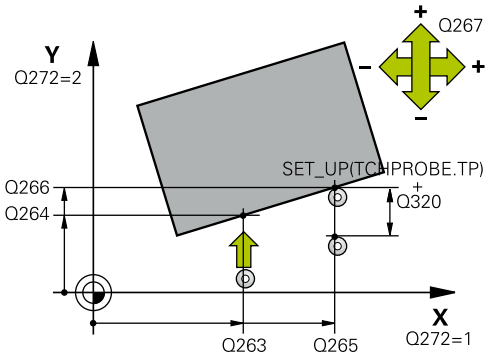
If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:

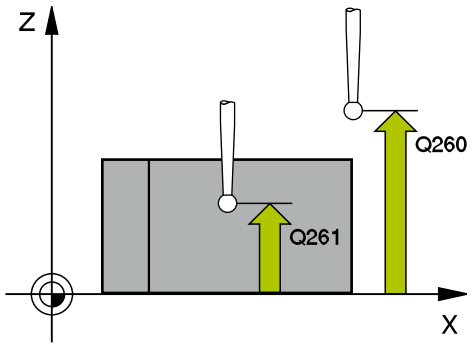
- If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
- If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET_UP**, and the ball-tip radius, even when probing in the tool axis direction. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999





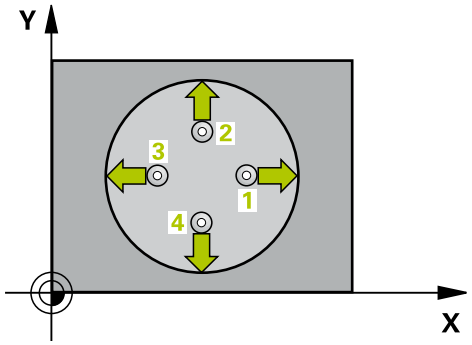
Example

```
5 TCH PROBE 420  MEASURE ANGLE
   Q263=+10   ;1ST POINT 1ST AXIS
   Q264=+10   ;1ST POINT 2ND AXIS
   Q265=+15   ;2ND PNT IN 1ST AXIS
   Q266=+95   ;2ND POINT 2ND AXIS
   Q272=1     ;MEASURING AXIS
   Q267=-1    ;TRAVERSE DIRECTION
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+10   ;CLEARANCE HEIGHT
   Q301=1     ;MOVE TO CLEARANCE
   Q281=1     ;MEASURING LOG
```

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR420.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**)

19.5 MEASURE HOLE (Circle 421, DIN/ISO: G421)

Cycle run



Touch probe cycle 421 measures the center point and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the SET_UP column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control 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 touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

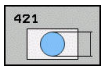
The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.

If parameter **Q330** references a turning tool, the following applies:

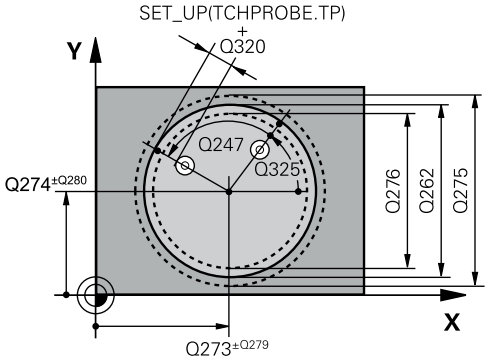
- Parameters **Q498** and **Q531** have to be defined
- The information in parameters **Q498**, **Q531**, for example from Cycle 800, has to match this information
- If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
- The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

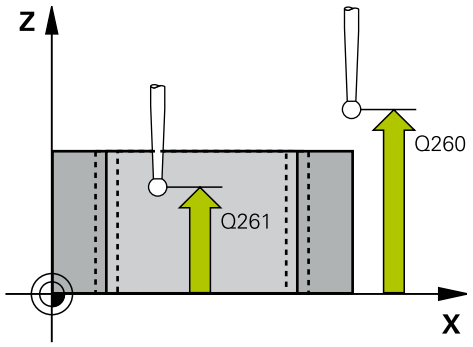
If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the hole in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: –360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999





Example

```
5 TCH PROBE 421  MEASURE HOLE
   Q273=+50   ;CENTER IN 1ST AXIS
   Q274=+50   ;CENTER IN 2ND AXIS
   Q262=75    ;NOMINAL DIAMETER
   Q325=+0    ;STARTING ANGLE
   Q247=+60   ;STEPPING ANGLE
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
```

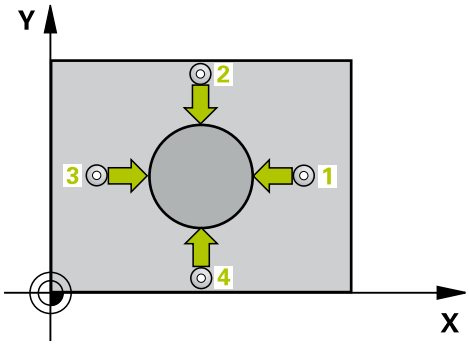
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q275 Maximum limit of size for hole?**: Maximum permissible diameter for the hole (circular pocket). Input range: 0 to 99999.9999
- **Q276 Minimum limit of size?**: Minimum permissible diameter for the hole (circular pocket). Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Definition of whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named 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 control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message

```
Q301=1       ;MOVE TO CLEARANCE
Q275=75.12   ;MAXIMUM LIMIT
Q276=74.95   ;MINIMUM LIMIT
Q279=0.1     ;TOLERANCE 1ST CENTER
Q280=0.1     ;TOLERANCE 2ND CENTER
Q281=1       ;MEASURING LOG
Q309=0       ;PGM STOP TOLERANCE
Q330=0       ;TOOL
Q423=4       ;NO. OF PROBE POINTS
Q365=1       ;TYPE OF TRAVERSE
Q498=0       ;REVERSE TOOL
Q531=0       ;ANGLE OF INCIDENCE
```

- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301**=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter **Q330**. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool Q498**=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle 800 and parameter **Reverse the tool Q498**=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter **Q330**. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle 800, **Angle of incidence? Q531**. Input range: –180° to +180°

19.6 MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422)

Cycle run



Touch probe cycle 422 measures the center point and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control 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 touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

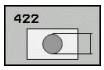
The smaller the stepping angle, the less accurately the control can calculate the dimensions of the stud. Minimum input value: 5°.

If parameter **Q330** references a turning tool, the following applies:

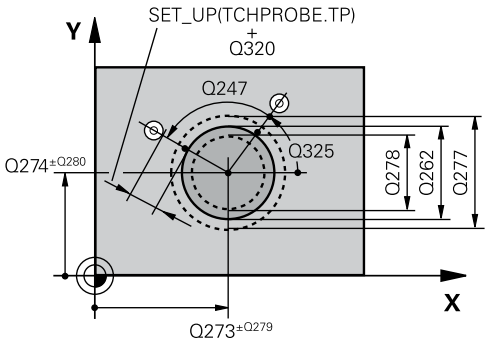
- Parameters **Q498** and **Q531** have to be defined
- The information in parameters **Q498**, **Q531**, for example from Cycle 800, has to match this information
- If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
- The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

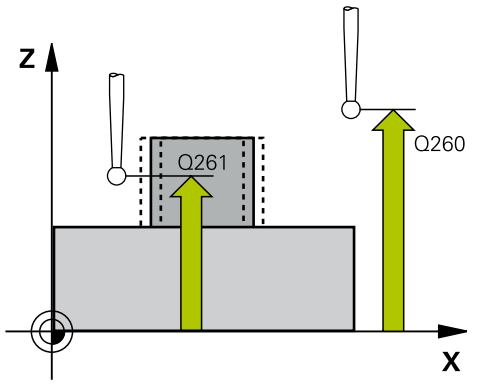
If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the diameter of the stud. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: –360.000 to 360.000
- **Q247 Intermediate stepping angle?** (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the machining direction (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.0000 to 120.0000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

```
5 TCH PROBE 422  MEAS. CIRCLE
    OUTSIDE
    Q273=+50    ;CENTER IN 1ST AXIS
    Q274=+50    ;CENTER IN 2ND AXIS
    Q262=75     ;NOMINAL DIAMETER
    Q325=+90    ;STARTING ANGLE
    Q247=+30    ;STEPPING ANGLE
    Q261=-5     ;MEASURING HEIGHT
    Q320=0      ;SET-UP CLEARANCE
    Q260=+10    ;CLEARANCE HEIGHT
    Q301=0      ;MOVE TO CLEARANCE
    Q277=35.15;MAXIMUM LIMIT
    Q278=34.9   ;MINIMUM LIMIT
```

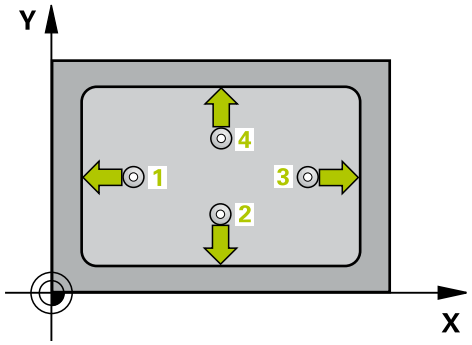
- **Q277 Maximum limit of size for stud?**: Maximum permissible diameter for the stud. Input range: 0 to 99999.9999
- **Q278 Minimum limit of size for stud?**: Minimum permissible diameter for the stud. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR422.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Tool number in the TOOL.T tool table
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points

```
Q279=0.05  ;TOLERANCE 1ST CENTER
Q280=0.05  ;TOLERANCE 2ND CENTER
Q281=1     ;MEASURING LOG
Q309=0     ;PGM STOP TOLERANCE
Q330=0     ;TOOL
Q423=4     ;NO. OF PROBE POINTS
Q365=1     ;TYPE OF TRAVERSE
Q498=0     ;REVERSE TOOL
Q531=0     ;ANGLE OF INCIDENCE
```

- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301**=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter **Q330**. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool Q498**=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle 800 and parameter **Reverse the tool Q498**=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter **Q330**. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle 800, **Angle of incidence? Q531**. Input range: –180° to +180°

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

Cycle run



Touch Probe Cycle 423 finds the center, length, and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

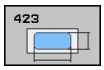
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

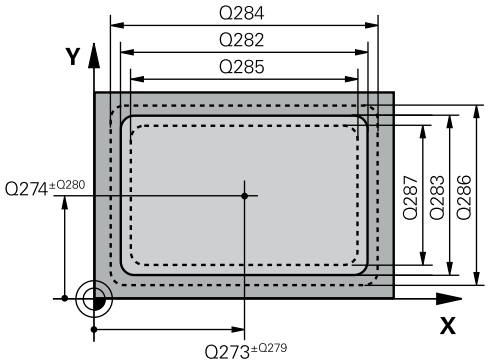
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

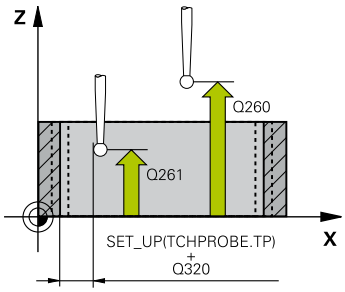
If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the pocket in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the pocket in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q282 1st side length (nominal value)?**: Pocket length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q283 2nd side length (nominal value)?**: Pocket length, parallel to the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q284 Max. size limit 1st side length?**: Maximum permissible length of the pocket. Input range: 0 to 99999.9999
- **Q285 Min. size limit 1st side length?**: Minimum permissible length of the pocket. Input range: 0 to 99999.9999
- **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the pocket. Input range: 0 to 99999.9999
- **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the pocket. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane. Input range: 0 to 99999.9999





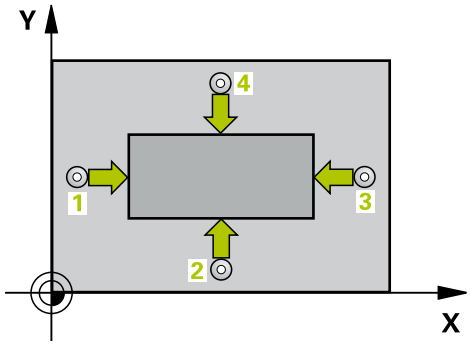
Example

5 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q282=80	;FIRST SIDE LENGTH
Q283=60	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q284=0	;MAX. LIMIT 1ST SIDE
Q285=0	;MIN. LIMIT 1ST SIDE
Q286=0	;MAX. LIMIT 2ND SIDE
Q287=0	;MIN. LIMIT 2ND SIDE
Q279=0	;TOLERANCE 1ST CENTER
Q280=0	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR423.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen.Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Tool number in the TOOL.T tool table

19.8 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 control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

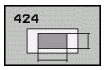
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

Please note while programming:

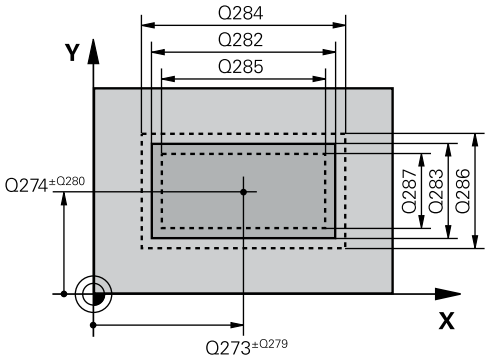
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

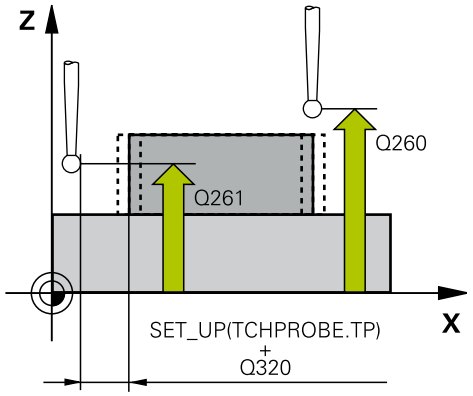
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q282 1st side length (nominal value)?**: Stud length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q283 2nd side length (nominal value)?**: Stud length, parallel to the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q284 Max. size limit 1st side length?**: Maximum permissible length of the stud. Input range: 0 to 99999.9999
- **Q285 Min. size limit 1st side length?**: Minimum permissible length of the stud. Input range: 0 to 99999.9999





Example

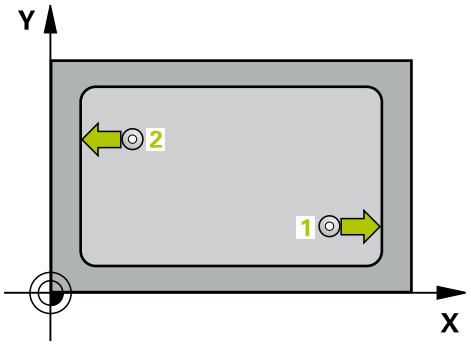
```
5 TCH PROBE 424  MEAS. RECTAN.
   OUTS.
   Q273=+50   ;CENTER IN 1ST AXIS
   Q274=+50   ;2ND CENTER 2ND AXIS
   Q282=75    ;FIRST SIDE LENGTH
   Q283=35    ;2ND SIDE LENGTH
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE
   Q284=75.1  ;MAX. LIMIT 1ST SIDE
   Q285=74.9  ;MIN. LIMIT 1ST SIDE
```

- **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the stud. Input range: 0 to 99999.9999
- **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the stud. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR424.TXT** by default in the directory that also contains the associated .h file.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

```
Q286=35     ;MAX. LIMIT 2ND SIDE
Q287=34.95;MIN. LIMIT 2ND SIDE
Q279=0.1    ;TOLERANCE 1ST CENTER
Q280=0.1    ;TOLERANCE 2ND CENTER
Q281=1      ;MEASURING LOG
Q309=0      ;PGM STOP TOLERANCE
Q330=0      ;TOOL
```

19.9 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 control makes a nominal-to-actual value comparison and saves the deviation value in a Q parameter.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.

3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.

4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

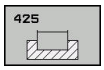
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Please note while programming:

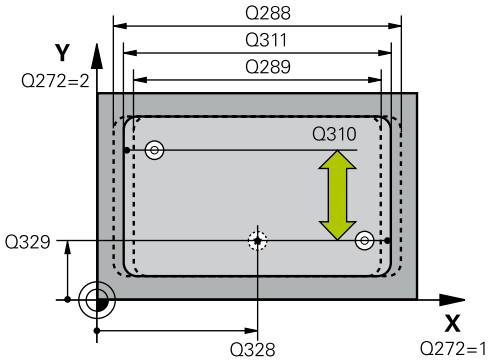
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

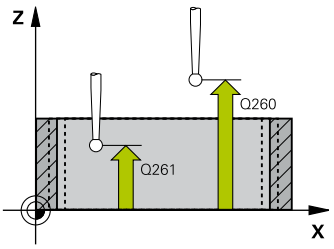
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q328 Starting point in 1st axis?** (absolute): Starting point for probing in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q329 Starting point in 2nd axis?** (absolute): Starting point for probing in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q310 Offset for 2nd measuremnt (+/-)?** (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the control does not offset the touch probe. Input range: –99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q311 Nominal length?** : Nominal value of the length to be measured. Input range: 0 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible length. Input range: 0 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible length. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: By default, the control will save the **log file named TCHPR425.TXT** in the directory that also contains the associated .h file.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.





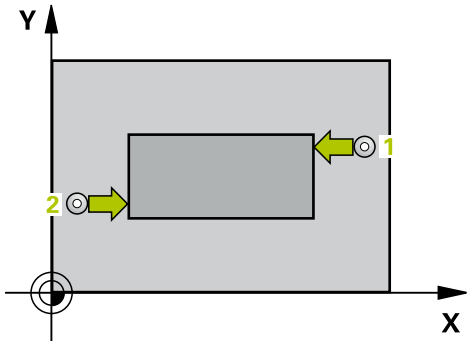
Example

```
5 TCH PROBE 425  MEASURE INSIDE WIDTH
   Q328=+75   ;STARTNG PNT 1ST AXIS
   Q329=-12.5 ;STARTNG PNT 2ND AXIS
   Q310=+0    ;OFFS. 2ND MEASUREMNT
   Q272=1     ;MEASURING AXIS
   Q261=-5    ;MEASURING HEIGHT
   Q260=+10   ;CLEARANCE HEIGHT
   Q311=25    ;NOMINAL LENGTH
   Q288=25.05;MAXIMUM LIMIT
   Q289=25    ;MINIMUM LIMIT
   Q281=1     ;MEASURING LOG
   Q309=0     ;PGM STOP TOLERANCE
   Q330=0     ;TOOL
   Q320=0     ;SET-UP CLEARANCE
   Q301=0     ;MOVE TO CLEARANCE
```

- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.
- **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points

19.10 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 control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). 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 touch point and probes it.

4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

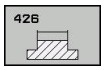
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Please note while programming:

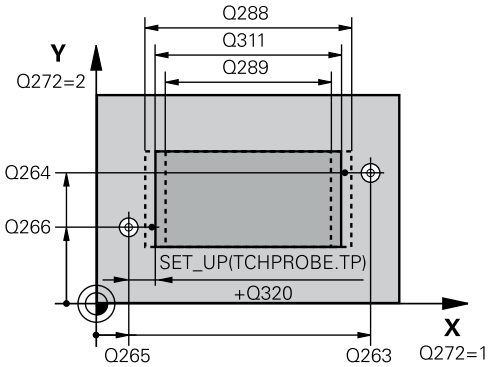
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

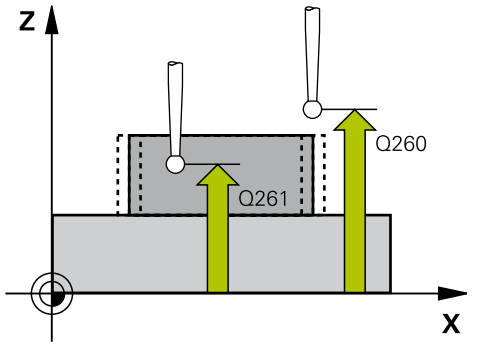
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q311 Nominal length?** : Nominal value of the length to be measured. Input range: 0 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible length. Input range: 0 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible length. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR426.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.





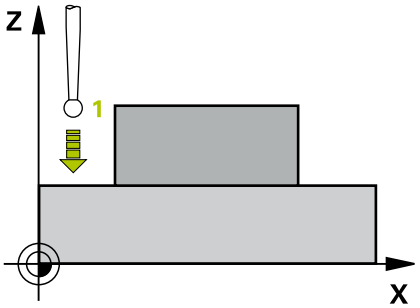
Example

```
5 TCH PROBE 426  MEASURE RIDGE
  WIDTH
  Q263=+50    ;1ST POINT 1ST AXIS
  Q264=+25    ;1ST POINT 2ND AXIS
  Q265=+50    ;2ND PNT IN 1ST AXIS
  Q266=+85    ;2ND PNT IN 2ND AXIS
  Q272=2      ;MEASURING AXIS
  Q261=-5     ;MEASURING HEIGHT
  Q320=0      ;SET-UP CLEARANCE
  Q260=+20    ;CLEARANCE HEIGHT
  Q311=45     ;NOMINAL LENGTH
  Q288=45     ;MAXIMUM LIMIT
  Q289=44.95;MINIMUM LIMIT
  Q281=1      ;MEASURING LOG
  Q309=0      ;PGM STOP TOLERANCE
  Q330=0      ;TOOL
```

- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

19.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)

Cycle run



The touch probe cycle 427 measures a coordinate in a selectable axis and saves the value in a Q parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Using Touch Probe Cycles", at touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.

3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

Parameter number	Meaning
Q160	Measured coordinate

Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).

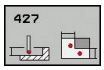
If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.

If parameter **Q330** references a turning tool, the following applies:

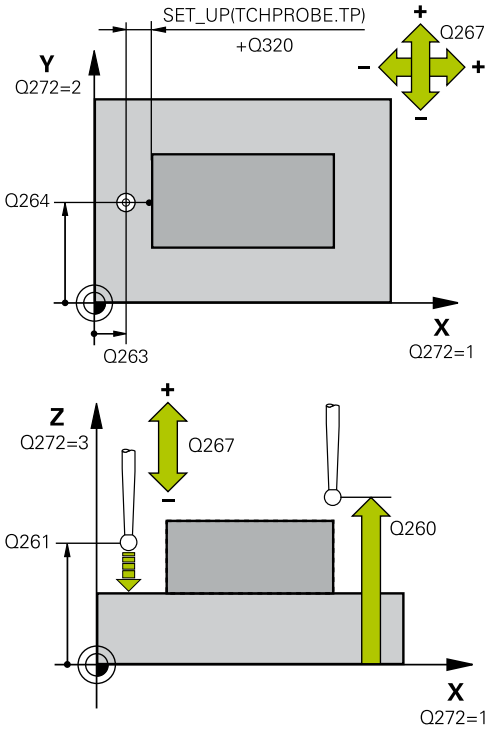
- Parameters **Q498** and **Q531** have to be defined
- The information in parameters **Q498**, **Q531**, for example from Cycle 800, has to match this information
- If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
- The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q281Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR427.TXT** in the same folder as the associated NC program.
  2: Interrupt the program run and display the measuring log on the screen.Resume the NC program run with **NC Start**.
- **Q288 Maximum limit of size?**: Maximum permissible measured value. Input range: –99999.9999 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible measured value. Input range: –99999.9999 to 99999.9999



Example

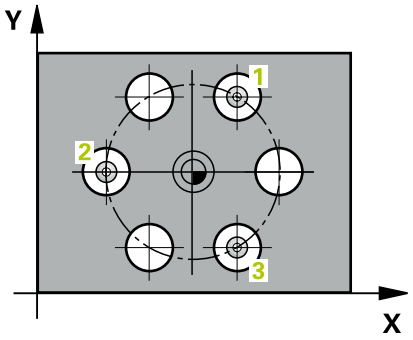
```
5 TCH PROBE 427  MEASURE
   COORDINATE
   Q263=+35    ;1ST POINT 1ST AXIS
   Q264=+45    ;1ST POINT 2ND AXIS
   Q261=+5     ;MEASURING HEIGHT
   Q320=0      ;SET-UP CLEARANCE
   Q272=3      ;MEASURING AXIS
   Q267=-1     ;TRAVERSE DIRECTION
   Q260=+20    ;CLEARANCE HEIGHT
   Q281=1      ;MEASURING LOG
   Q288=5.1    ;MAXIMUM LIMIT
   Q289=4.95   ;MINIMUM LIMIT
   Q309=0      ;PGM STOP TOLERANCE
   Q330=0      ;TOOL
   Q498=0      ;REVERSE TOOL
   Q531=0      ;ANGLE OF INCIDENCE
```

- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter **Q330**. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool Q498**=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle 800 and parameter **Reverse the tool Q498**=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter **Q330**. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle 800, **Angle of incidence? Q531**. Input range: –180° to +180°

19.12 MEAS. BOLT HOLE (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 control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), to the programmed center point of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt circle diameter

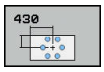
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

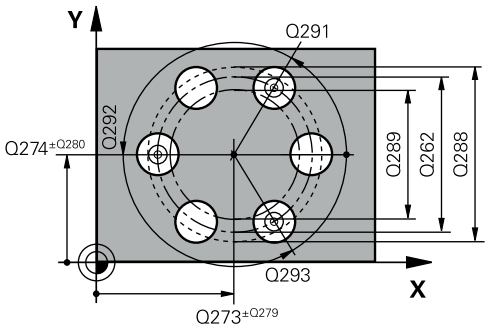
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

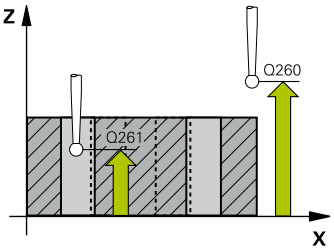
Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range: 0 to 99999.9999
- **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range: –360.0000 to 360.0000
- **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range: –360.0000 to 360.0000
- **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range: –360.0000 to 360.0000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: –99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999





Example

```
5 TCH PROBE 430  MEAS. BOLT HOLE
  CIRC
  Q273=+50   ;CENTER IN 1ST AXIS
  Q274=+50   ;CENTER IN 2ND AXIS
  Q262=80    ;NOMINAL DIAMETER
  Q291=+0    ;ANGLE OF 1ST HOLE
  Q292=+90   ;ANGLE OF 2ND HOLE
  Q293=+180  ;ANGLE OF 3RD HOLE
  Q261=-5    ;MEASURING HEIGHT
  Q260=+10   ;CLEARANCE HEIGHT
  Q288=80.1  ;MAXIMUM LIMIT
```

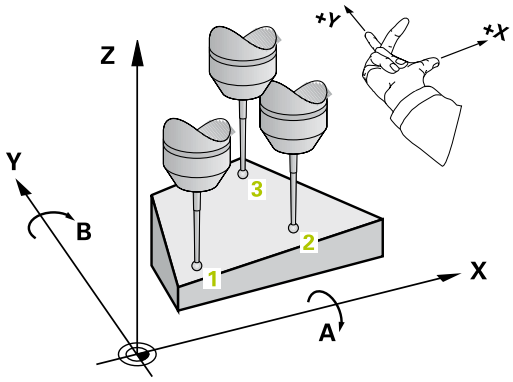
- **Q289 Minimum limit of size?**: Minimum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR430.TXT** in the directory that also contains the associated NC program
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 710). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

```
Q289=79.9  ;MINIMUM LIMIT
Q279=0.15  ;TOLERANCE 1ST CENTER
Q280=0.15  ;TOLERANCE 2ND CENTER
Q281=1     ;MEASURING LOG
Q309=0     ;PGM STOP TOLERANCE
Q330=0     ;TOOL
```

19.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)

Cycle run

Touch probe cycle 431 finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.



1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 595), at the programmed touch point **1** and measures the first point of the plane. The control 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 touch point **2** and measures the actual value of the second touch point in the plane.

3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.

4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to **Q175**	Measured values in the touch probe axis (first to third measurement)

Please note while programming:

NOTICE

Danger of collision!

If you save the values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA**=0; **SPB**=0; **SPC**=0, there are multiple solutions in which the tilting axes are at 0.

- Make sure to program **SYM** (**SEQ**) + or **SYM** (**SEQ**) -

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

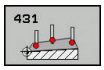
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control can only calculate the angular values if the three measuring points are not positioned on a straight line.

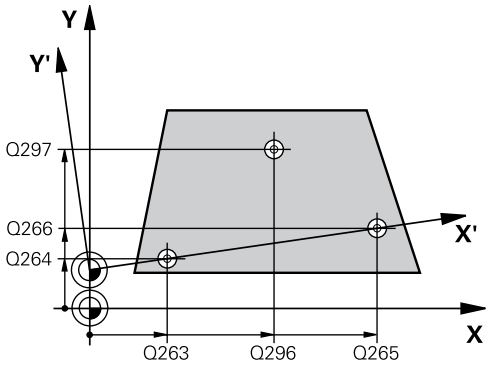
The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points you also specify the direction of the principal axis when tilting the working plane.

The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

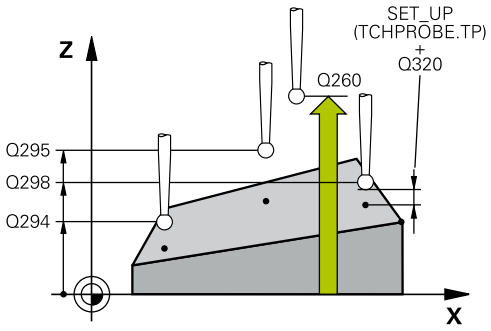
Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: –99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999



- **Q295 2nd measuring point in 3rd axis?** (absolute): Coordinate of the second touch point in the touch probe axis. Input range: –99999.9999 to 99999.9999
- **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q298 3rd measuring point in 3rd axis?** (absolute): Coordinate of the third touch point in the touch probe axis. Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the directory that also contains the associated NC program
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.



Example

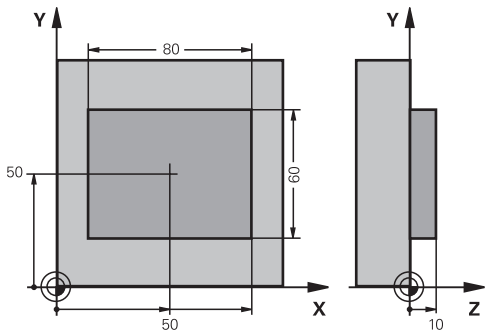
```
5 TCH PROBE 431  MEASURE PLANE
   Q263=+20   ;1ST POINT 1ST AXIS
   Q264=+20   ;1ST POINT 2ND AXIS
   Q294=-10   ;1ST POINT 3RD AXIS
   Q265=+50   ;2ND PNT IN 1ST AXIS
   Q266=+80   ;2ND PNT IN 2ND AXIS
   Q295=+0    ;2ND PNT IN 3RD AXIS
   Q296=+90   ;3RD PNT IN 1ST AXIS
   Q297=+35   ;THIRD POINT 2ND AXIS
   Q298=+12   ;3RD PNT IN 3RD AXIS
   Q320=0     ;SET-UP CLEARANCE
   Q260=+5    ;CLEARANCE HEIGHT
   Q281=1     ;MEASURING LOG
```

19.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program run

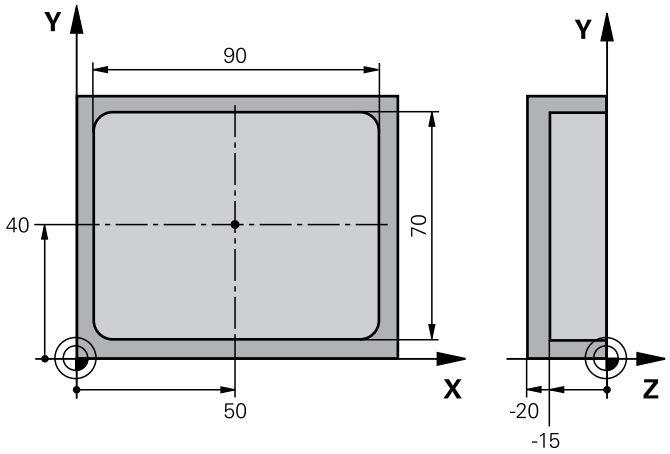
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



0 BEGIN PGM BEAMS MM	
1 TOOL CALL 69 Z	Tool call: roughing
2 L Z+100 R0 FMAX	Retract the tool
3 FN 0: Q1 = +81	Rectangle length in X (roughing dimension)
4 FN 0: Q2 = +61	Rectangle length in Y (roughing dimension)
5 CALL LBL 1	Call subprogram for machining
6 L Z+100 R0 FMAX	Retract the tool
7 TOOL CALL 99 Z	Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS.	Measure the rough-milled rectangle
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+50 ;CENTER IN 2ND AXIS	
Q282=80 ;FIRST SIDE LENGTH	Nominal length in X (final dimension)
Q283=60 ;2ND SIDE LENGTH	Nominal length in Y (final dimension)
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=0 ;MAX. LIMIT 1ST SIDE	Input values for tolerance checking not required
Q285=0 ;MIN. LIMIT 1ST SIDE	
Q286=0 ;MAX. LIMIT 2ND SIDE	
Q287=0 ;MIN. LIMIT 2ND SIDE	
Q279=0 ;TOLERANCE 1ST CENTER	
Q280=0 ;TOLERANCE 2ND CENTER	
Q281=0 ;MEASURING LOG	No measuring log transmission
Q309=0 ;PGM STOP TOLERANCE	Do not output an error message
Q330=0 ;TOOL	No tool monitoring
9 FN 2: Q1 = +Q1 - +Q164	Calculate length in X including the measured deviation
10 FN 2: Q2 = +Q2 - +Q165	Calculate length in Y including the measured deviation
11 L Z+100 R0 FMAX	Retract the touch probe

12 TOOL CALL 1 Z S5000	Tool call: finishing
13 CALL LBL 1	Call subprogram for machining
14 L Z+100 R0 FMAX M2	Retract the tool, end program
15 LBL 1	Subprogram with rectangular stud machining cycle
16 CYCL DEF 213 STUD FINISHING	
Q200=20 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE MILLING	
Q203=+10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=Q1 ;FIRST SIDE LENGTH	Length in X variable for roughing and finishing
Q219=Q2 ;2ND SIDE LENGTH	Length in Y variable for roughing and finishing
Q220=0 ;CORNER RADIUS	
Q221=0 ;ALLOWANCE IN 1ST AXS	
17 CYCL CALL M3	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	

Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call: touch probe
2 L Z+100 R0 FMAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+40 ;CENTER IN 2ND AXIS	
Q282=90 ;FIRST SIDE LENGTH	Nominal length in X
Q283=70 ;2ND SIDE LENGTH	Nominal length in Y
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+20 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=90.15 ;MAX. LIMIT 1ST SIDE	Maximum limit in X
Q285=89.95 ;MIN. LIMIT 1ST SIDE	Minimum limit in X
Q286=70.1 ;MAX. LIMIT 2ND SIDE	Maximum limit in Y
Q287=69.9 ;MIN. LIMIT 2ND SIDE	Minimum limit in Y
Q279=0.15 ;TOLERANCE 1ST CENTER	Permissible position deviation in X
Q280=0.1 ;TOLERANCE 2ND CENTER	Permissible position deviation in Y
Q281=1 ;MEASURING LOG	Save measuring log to a file
Q309=0 ;PGM STOP TOLERANCE	Do not display an error message in case of a tolerance violation
Q330=0 ;TOOL	No tool monitoring
4 L Z+100 R0 FMAX M2	Retract the tool, end program
5 END PGM BSMEAS MM	

20

Touch Probe Cycles: Special Functions

20.1 Fundamentals

Overview

NOTICE

Danger of collision!

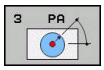
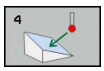
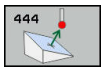
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

The control provides cycles for the following special purposes:

Soft key	Cycle	Page
	3 **MEASURING** Measuring cycle for defining OEM cycles	755
	4 **MEASURING IN 3-D** Measuring any position	757
	444 **PROBING IN 3-D** Measuring any position	759
	441 **FAST PROBING** Measuring cycle for defining various touch probe parameters	764

20.2 MEASURING (Cycle 3)

Cycle run

Touch probe cycle 3 measures any position on the workpiece in a selectable probing direction. Unlike other measuring cycles, Cycle 3 enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value **MB** after determining the measured value.

1 The touch probe moves from the current position at the specified feed rate in the defined probing direction. Use polar angles to define the probing direction in the cycle.

2 After the control has saved the position, the touch probe stops. The control saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The control does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.

3 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

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.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other measuring cycles, do not apply in Touch Probe Cycle 3.

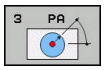
Remember that the control always writes to four successive Q parameters.

If the control was not able to determine a valid touch point, the NC program is run without error message. In this case the control assigns the value –1 to the fourth result parameter so that you can deal with the error yourself.

The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.

With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.

Cycle parameters



- **Parameter number for result?**: Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to 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 traverse distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range: –99999.9999 to 99999.9999
- **Feed rate measurement**: Enter the feed rate for measurement in mm/min. Input range: 0 to 3000.000
- **Maximum retraction distance?**: Traverse distance in the direction opposite the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input range: 0 to 99999.9999
- **Reference system? (0=ACT/1=REF)**: Define whether the probing direction and measuring result should reference the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):
  0: Probe in the current system and save the measuring result to the **ACT** system
  1: Probe in the machine-based REF system. Save the measuring result in the REF system.
- **Error mode? (0=OFF/1=ON)**: Specify whether the control is to issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the fourth result parameter and continues the cycle:
  0: Display error message
  1: Do not display error message

Example

```
4 TCH PROBE 3.0  MEASURING
5 TCH PROBE 3.1  Q1
6 TCH PROBE 3.2  X ANGLE:  +15
7 TCH PROBE 3.3  ABST +10  F100 MB1
    REFERENCE SYSTEM: 0
8 TCH PROBE 3.4  ERRORMODE1
```

20.3 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 control 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 probing distance and probing feed rate directly. You can also define the distance by which the touch probe retracts after determining the measured value.

1 The control moves the touch probe 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 control has saved the position, the control stops the probe movement. The control 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 control retracts the touch probe in the direction opposite to the direction of probing. You define the traverse distance in parameter **MB**—the touch probe is moved to a point no farther than the starting point.

Please note while programming:

NOTICE

Danger of collision!

If the control was not able to determine a valid touch point, the 4th result parameter will have the value -1. The control does **not** interrupt the program run!

- Make sure that all touch points can be reached.

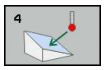
This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.

Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position!

Remember that the control always writes to four successive Q parameters.

Cycle parameters



- **Parameter number for result?**: Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input range: 0 to 1999
- **Relative measuring path in X?**: X component of the direction vector defining the direction in which the touch probe is to move. Input range: –99999.9999 to 99999.9999
- **Relative measuring path in Y?**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range: –99999.9999 to 99999.9999
- **Relative measuring path in Z?**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range: –99999.9999 to 99999.9999
- **Maximum measuring range?**: Enter the maximum traverse distance from the starting point by which the touch probe may move along the direction vector. Input range: –99999.9999 to 99999.9999
- **Feed rate measurement**: Enter the feed rate for measurement in mm/min. Input range: 0 to 3000.000
- **Maximum retraction distance?**: Traverse distance in the direction opposite the probing direction, after the stylus was deflected. Input range: 0 to 99999.9999
- **Reference system? (0=ACT/1=REF)**: Specify whether the result of probing is to be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**):
  0: Save measuring result to the **ACT** system
  1: Save measuring result to the **REF** system

Example

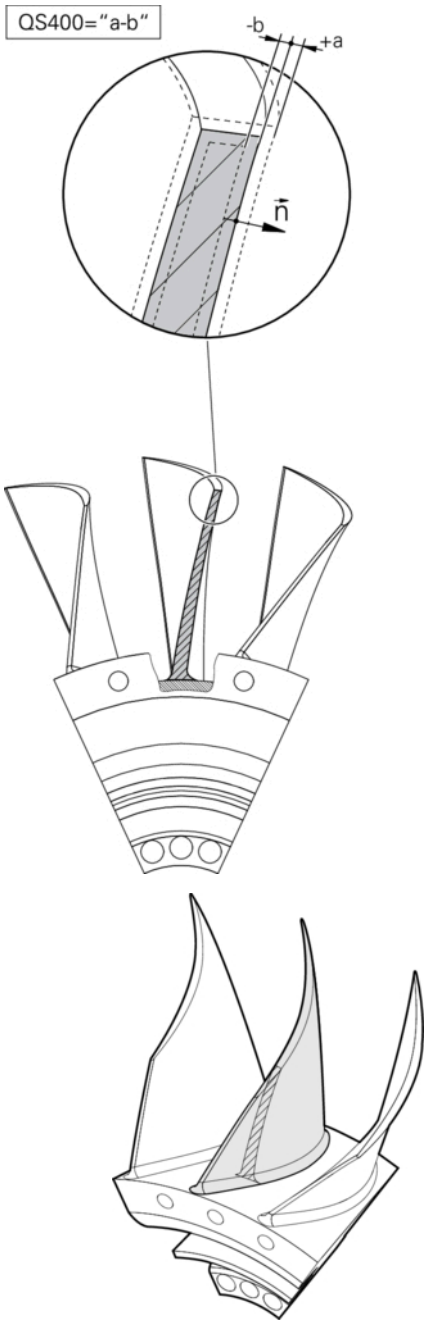
```
4 TCH PROBE 4.0  MEASURING IN 3-D
5 TCH PROBE 4.1  Q1
6 TCH PROBE 4.2  IX-0.5 IY-1 IZ-1
7 TCH PROBE 4.3 ABST+45 F100 MB50
    REFERENCE SYSTEM:0
```

20.4 PROBING IN 3-D (Cycle 444, ISO: G444)

Cycle run

Cycle 444 checks one specific point on the surface of a component. This cycle is used, for example, to measure free-form surfaces of molded parts. It can be determined whether a point on the surface of the component lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle 444 probes any point in three dimensions, and determines the deviation to a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface, and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the Q parameters listed below.



Cycle run

1 Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table \tchprobe.tp) + **Q320** Pre-positioning takes a clearance height into account. For more information on the probing logic, see "Executing touch probe cycles", Page 595

2 The touch probe then moves to the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.

3 After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.

4 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Q parameters

The control stores the probing results in the following parameters:

Parameter number	Meaning
Q151	Measured position in principal axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in reference axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3-D deviation ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the principal, secondary, and tool axes as well as the 3-D deviation. The control saves the log in the same folder in which the *.h file is located (as long as no path has been configured for FN16).

The log contains the following data on the principal, secondary, and tool axes:

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- If a tolerance **QS400** was defined: Upper and lower allowances are output, as well as the determined deviation along the normal vector
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")

Please note while programming!

Depending on the setting of the optional **chkTiltingAxes** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

In order to obtain exact results from the touch probe being used, you should perform a 3-D calibration before executing Cycle 444. Software option 92, 3D-ToolComp, is required for 3-D calibration.

Cycle 444 generates a measuring log in HTML format.

An error message is output if a mirroring (Cycle 8) or scaling (Cycle 11, 26) is active before Cycle 444 is run.

For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is even possible if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.

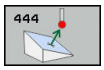
If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.

Cycle 444 references all coordinates to the input system.

The control writes the measured values to return parameters, see "Cycle run", Page 759.

The workpiece status good/rework/scrap is set via Q parameter **Q183**, independent of parameter **Q309** (see "Cycle run", Page 759).

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: –99999.9999 to 99999.9999
- **Q581 Surface-normal in ref. axis?** Enter here the surface normal in the direction of the principal axis. The surface normal of a point is normally output by a CAD/CAM system. Input range: –10 to 10
- **Q582 Surface-normal in minor axis?** Enter here the surface normal in the direction of the secondary axis. The surface normal of a point is normally output by a CAD/CAM system. Input range: –10 to 10
- **Q583 Surface-normal in tool axis?** Enter here the surface normal in the direction of the tool axis. The surface normal of a point is normally output by a CAD/CAM system. Input range: –10 to 10
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: –99999.9999 to 99999.9999

Example

```
4 TCH PROBE 444 PROBING IN 3-D
  Q263=+0     ;1ST POINT 1ST AXIS
  Q264=+0     ;1ST POINT 2ND AXIS
  Q294=+0     ;1ST POINT 3RD AXIS
  Q581=+1     ;NORMAL IN REF. AXIS
  Q582=+0     ;NORMAL IN MINOR AXIS
  Q583=+0     ;NORMAL IN TOOL AXIS
  Q320=+0     ;SAFETY CLEARANCE
  Q260=100    ;CLEARANCE HEIGHT
  QS400="1-1";TOLERANCE
  Q309=+0     ;ERROR REACTION
```

- **QS400 Tolerance value?** Specify a tolerance range that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normals. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by **Q581** through **Q583**, and the nominal coordinate is defined by **Q263**, **Q264**, and **Q294**.) The tolerance value is distributed over the axes, depending on the normal vector:
  Example: QS400 ="0.4-0.1" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate –0.1. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate - 0.1".
  Example: QS400 ="0.4" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate".
  Example: QS400 ="-0.1" means: upper allowance = nominal coordinate, lower allowance = nominal coordinate –0.1. The following tolerance range thus results for the cycle: "nominal coordinate" to "nominal coordinate –0.1".
  Example: QS400 =" " means: No tolerance band.
  Example: QS400 ="0" means: No tolerance band.
  Example: QS400 ="0.1+0.1" means: No tolerance band.
- **Q309 Reaction to tolerance error?** Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not the interrupt program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the value of the determined actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts the NC program run. However, there will be no error message if the value of the determined actual coordinate is higher than that of the nominal coordinate.

20.5 FAST PROBING (Cycle 441, DIN/ISO: G441)

Cycle run

You can use the touch probe cycle 441 to globally specify various touch probe parameters (e.g. the positioning feed rate) for all subsequently used touch probe cycles.

Please note while programming:

Your machine tool builder may have limited the feed rate additionally. You can set the maximum absolute feed rate in the **maxTouchFeed** machine parameter (no. 122602).

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The purpose of Cycle 441 is to set parameters for probing cycles. In this cycle, no machine movements will be performed.

END PGM, **M2**, **M30** resets the global settings of Cycle 441.

Cycle parameter **Q399** depends on your machine configuration. Orienting the touch probe system via the NC program must be enabled by your machine tool builder.

Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397**=1.

Cycle parameters

- **Q396 Positioning feed rate?**: Define the feed rate at which the touch probe will be moved to the specified positions. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q397 Pre-pos. at machine's rapid?**: Define whether the control, when pre-positioning the touch probe, traverses at **FMAX** feed rate (machine's rapid traverse):
  0: Pre-position with feed rate from **Q396**
  1: Pre-position with machine's rapid traverse **FMAX**. Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397**=1. Your machine tool builder may have limited the feed rate additionally. You can set the maximum absolute feed rate in the **maxTouchFeed** machine parameter (no. 122602).
- **Q399 Angle tracking (0/1)?**: Define whether the control orients the touch probe before each probing operation:
  0: No orientation
  1: Orient spindle before each probing operation (improves the accuracy)
- **Q400 Automatic interruption?** Define whether the control interrupts the program run and outputs the measuring results on the screen following a measuring cycle for automatic workpiece measurement:
  0: Do not interrupt program run even if, in the specific probing cycle, the output of measuring results on the screen is selected
  1: Interrupt program run and output measuring results on the screen. You can then resume the NC program run with **NC Start**.

Example

```
5 TCH PROBE 441  FAST PROBING
    Q 396=3000;POSITIONING FEED RATE
    Q 397=0    ;SELECT FEED RATE
    Q 399=1    ;ANGLE TRACKING
    Q 400=1    ;INTERRUPTION
```

20.6 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 control cannot provide precise measuring results.

Always calibrate a touch probe in the following cases:

- Commissioning
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, due to heating up of the machine
- Change of active tool axis

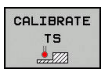
The control assumes the calibration values for the active probe system directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.

During calibration, the control 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 control provides calibration cycles for calibrating the length and the radius:

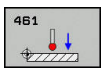
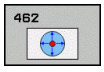
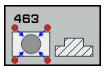
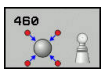
Proceed as follows:

- Press the **TOUCH PROBE** key.

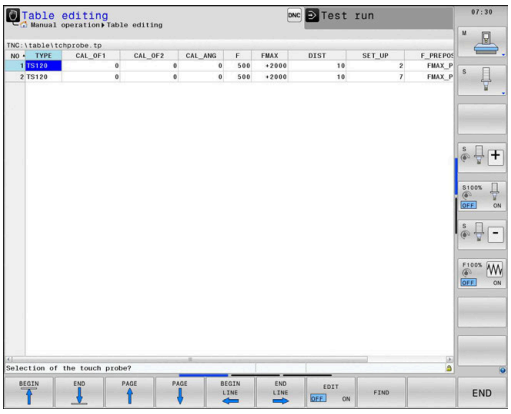


- Press the **CALIBRATE TS** soft key
- Select the calibration cycle

Calibration cycles provided by the control

Soft key	Function	Page
	Calibrating the length	768
	Measure the radius and the center offset using a calibration ring	770
	Measuring the radius and the center offset using a stud or a calibration pin	773
	Measure the radius and the center offset using a calibration sphere	776

20.7 Displaying calibration values



The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (principal axis) and **CAL_OF2** (secondary axis). 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 an NC 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 control saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC:*.

Ensure that the tool number in the tool table and the touch-probe number of the touch-probe table match. This is regardless of whether you want to use a touch-probe cycle in automatic mode or in **Manual operation** mode.

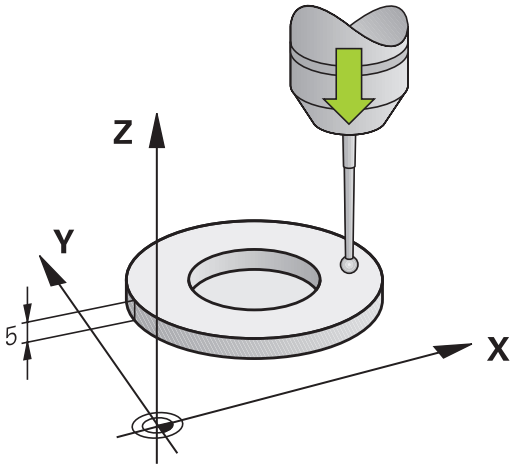
For more information, see Chapter Touch-probe table

20.8 TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461)

Cycle run

Before starting the calibration cycle, you must set the preset in the spindle axis so that Z=0 on the machine table; you must also pre-position the touch probe above 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 an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.



1 The control orients the touch probe to the angle **CAL_ANG** specified in the touch probe table (only if your touch probe can be oriented).
2 The control probes from the current position in the negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
3 The control then retracts the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the starting position.

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

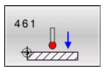
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

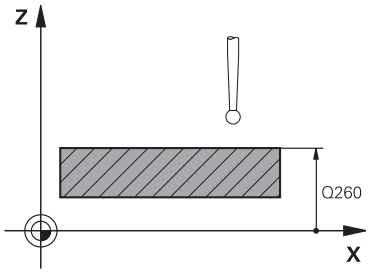
The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.



- **Q434 Preset for length?** (absolute): Preset for the length (e.g. height of the calibration ring). Input range: –99999.9999 to 99999.9999



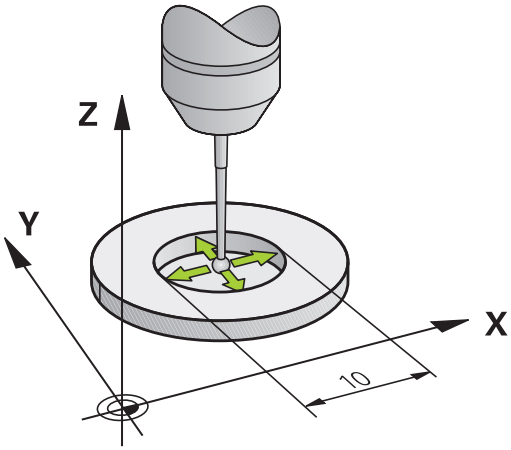
Example

```
5 TCH PROBE 461  TS CALIBRATION OF
    TOOL LENGTH
    Q434=+5     ;PRESET
```

20.9 CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462)

Cycle run

Refer to your machine manual!



Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

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 an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. 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): Probing operation: see "Orientation possible in two directions").

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer. Please refer to the machine manual.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

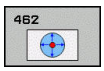
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

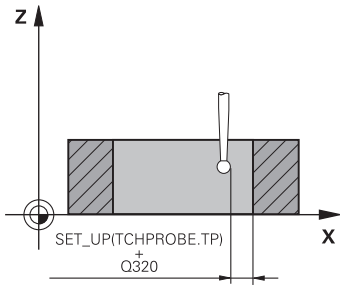
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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.



- **Q407 RING RADIUS** Enter the radius of the calibration ring. Input range: 0 to 9.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q380 Ref. angle in ref. axis?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: 0 to 360.0000



Example

```
5 TCH PROBE 462  CALIBRATION OF A
    TS IN A RING
    Q407=+5     ;RING RADIUS
    Q320=+0     ;SET-UP CLEARANCE
    Q423=+8     ;NO. OF PROBE POINTS
    Q380=+0     ;REFERENCE ANGLE
```

20.10 TS CALIBRATION ON STUD (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 control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

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 an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. 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): Probing operation: see "Orientation possible in two directions").

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer. Please refer to the machine manual.

The orientation property of your touch probe is already predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

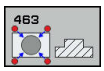
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

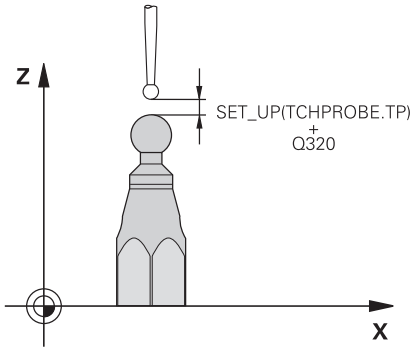
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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.



- **Q407 Radius of calibr. stud?**: Diameter of the calibration ring or pin. Input range: 0 to 99.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q380 Ref. angle in ref. axis?** (absolute): Angle between the principal axis of the working plane and the first touch point. Input range: 0 to 360.0000



Example

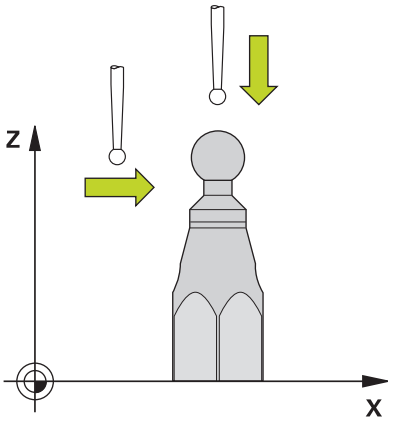
```
5 TCH PROBE 463  TS CALIBRATION ON STUD
   Q407=+5      ;STUD RADIUS
   Q320=+0      ;SET-UP CLEARANCE
   Q301=+1      ;MOVE TO CLEARANCE
   Q423=+8      ;NO. OF PROBE POINTS
   Q380=+0      ;REFERENCE ANGLE
```

20.11 CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460)

Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration sphere. 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 sphere.

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

It is also possible to capture 3-D calibration data. Option 92, 3D-ToolComp, is required for this purpose. 3-D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3-D calibration data are stored under TNC: \system\3D-ToolComp*. The DR2TABLE column of the tool table references the 3DTC table. The 3-D calibration data are then taken into account when probing. This 3-D calibration is necessary if you want to achieve a very high accuracy with Cycle 444 3-D Probing (see "PROBING IN 3-D (Cycle 444, ISO: G444)", Page 759).

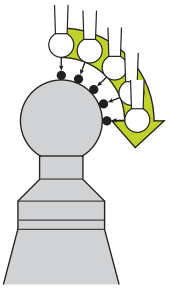


Cycle run

The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

Radius calibration Q433=0

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3 The first movement is in the plane, depending on the reference angle (**Q380**).
4 The control then positions the touch probe in touch-probe axis.
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
6 Once the equator has been determined, the radius calibration begins
7 Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.



Radius and length calibration Q433=1

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3 The first movement is in the plane, depending on the reference angle (**Q380**).
4 The control then positions the touch probe in touch-probe axis.
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
6 Once the equator has been determined, the radius calibration begins
7 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.
8 The control determines the length of the touch probe at the north pole of the calibration sphere.
9 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

The setting in parameter **Q455** specifies whether you can perform an additional 3-D calibration.

3-D calibration Q455= 1...30

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 After calibration of the radius and length, the control retracts the touch probe in touch-probe axis. Then the control positions the touch probe above the north pole.
3 The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are thus determined
4 You can specify the number of probing points between the north pole and the equator. This number depends on input parameter **Q455**. A value between 1 and 30 can be programmed. If you program **Q455**=0, no 3-D calibration will be performed.
5 The deviations determined during the calibration are stored in a 3DTC table.
6 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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 an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

Before a cycle definition you must program a tool call to define the touch-probe axis.

Pre-position the touch probe so that it is located approximately above the center of the calibration sphere.

If you program **Q455**=0, the control will not perform a 3-D calibration.

If you program **Q455**=1 to 30, the control will perform a 3-D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles. If you use Cycle 444, you should first perform a 3-D calibration.

If you program **Q455**=1 to 30, a table will be stored under TNC:\system\3D-ToolComp*.

If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.

If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.

- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q380 Ref. angle in ref. axis?** (absolute): Enter the reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000
- **Q433 Calibrate length (0/1)?**: Define whether the control is to calibrate the touch probe length after radius calibration, as well:
  0: Do not calibrate touch probe length
  1: Calibrate touch probe length
- **Q434 Preset for length?** (absolute): Coordinate of the calibration sphere center. This value must only be defined if length calibration is to be carried out. Input range: –99999.9999 to 99999.9999
- **Q455 No. of points for 3-D calibrtn.?** Enter the number of touch points for 3-D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3-D calibration. During 3-D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3-D calibration. Input range: 1 to 30

Example

```
5 TCH PROBE 460  CALIBRATION OF TS
  ON A SPHERE
  Q407=12.5  ;SPHERE RADIUS
  Q320=0     ;SET-UP CLEARANCE
  Q301=1     ;MOVE TO CLEARANCE
  Q423=4     ;NO. OF PROBE POINTS
  Q380=+0    ;REFERENCE ANGLE
  Q433=0     ;CALIBRATE LENGTH
  Q434=-2.5  ;PRESET
  Q455=15    ;NO. POINTS 3-D CAL.
```

21

VSC: camera-based setup control (option 136)

21.1 Camera-based Visual Setup Control VSC (Option 136)

Fundamentals

In order to use Visual Setup Control you need the following components:

- Software: Option 136 Visual Setup Control (VSC)
- Hardware: Camera system from HEIDENHAIN

Application

Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Camera-based setup control (Option 136 Visual Setup Control) can be used to monitor the current setup situation before and during machining and compare it to a safe desired condition. After setup, simple cycles for automatic monitoring are available.

A camera system generates reference images of the current working space. With Cycles 600 **GLOBAL WORKING SPACE** or 601 **LOCAL WORKING SPACE**, the control produces an image of the working space and compares the image with previously prepared reference images. These cycles can highlight irregularities in the working space. The operator decides whether the NC program is interrupted in the event of an error or continues to run.

Using VSC offers the following advantages:

- The control can recognize elements (e.g. tools, fixtures, etc.) that are in the working space once the program has started
- If you always want to clamp a workpiece at the same position (e.g. hole at top right), the control can check the setup situation
- For documentation purposes you can generate an image of the current workspace (e.g. of a clamping situation that is rarely used)

Further information: User's Manual for Setup, Testing and Running NC programs

Terms

The following terms are used in connection with VSC:

Term	Explanation
Reference image	A reference image shows a situation in the working space that you regard as safe. Therefore only create reference images showing safe, non-hazardous situations.
Mid-value image	The control creates a mid-value image taking into account all reference images. The control compares new images with the mid-value image as part of evaluation.
Images with error	If you generate an image showing a poor situation (such as an incorrectly clamped workpiece), you can generate an "image of error" It is not advisable to highlight an error image as a reference image.
Monitoring area	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.
Error	Area on an image containing a deviation from the desired position. Errors always refer to the image for which they were saved (image of error) or to the image most recently evaluated.
Monitoring phase	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.

Managing 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

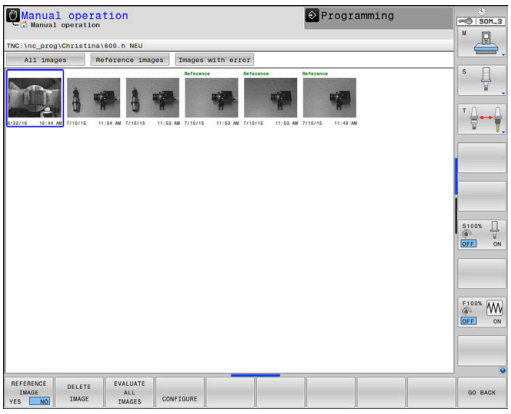
- Press the **MONITORING DATA MANAGEMENT** soft key
- > The control shows a list of the NC programs being monitored.

- Press the **OPEN** soft key
- > The control shows a list of monitoring points.
- Edit the desired data

Selecting data

You can select the buttons with the mouse. These interfaces make it easier to search and show results in a manageable way.

- **All images**: Display all images for this monitoring file
- **Reference images**: Only display reference images
- **Images with error**: Display all images where you have highlighted an error



Features of the monitoring data management

Soft key	Function
REFERENCE IMAGE YES NO	Mark selected image as a reference image Please note: A reference image shows a situation in the working space that you regard as safe. All reference images are taken into account for the evaluation. If you add or remove an image as a reference image, this has an effect on the results of image evaluation.
DELETE IMAGE	Delete image currently selected
EVALUATE ALL IMAGES	Carry out automatic image evaluation The control carries out an image evaluation according to the reference images and the monitoring areas.
CONFIGURE	Change monitoring area or highlight an error
GO BACK	Go back to the previous screen If you change the configuration, the control carries out an image evaluation.

Overview

The control provides two cycles you can use to define visual setup control in **Programming** mode of operation:

TOUCH PROBE

- The soft-key row shows all available touch probe functions divided into groups.

MONITORING WITH CAMERA

- Press the **MONITORING WITH CAMERA** soft key.

Soft key	Cycle	Page
	600 **GLOBAL WORKING SPACE**	790
	601 **LOCAL WORKING SPACE**	795

Configuration

You can change your settings regarding the monitoring area and errors at any time. When you press the **CONFIGURE** soft key, the soft-key row is switched and you can change your settings.

Soft key	Function
CONFIGURE	Change settings for the monitoring area and sensitivity If you make a change in this menu, the results of the image evaluation may change.
DRAW AREA	Draw new monitoring area If you add a new monitoring area, or change/ delete monitoring areas already set, this will have an effect on the image evaluation. The same monitoring area applies to all reference images.
DRAW ERROR	Draw new error
EVALUATE IMAGE	The control checks if or how the new settings affect this image
EVALUATE ALL IMAGES	The control checks if or how the new settings affect all images
SHOW AREAS	The control shows all drawn monitoring areas
SHOW COMPARISON	The control compares the momentary image with the mean image
SAVE AND GO BACK	Save current image and return to the previous screen If you change the configuration, the control carries out an image evaluation.
GO BACK	Discard images and return to the previous screen

Drawing the monitoring area or error area

Proceed as follows:

- ▶ Press the appropriate soft key, e.g. **DRAW AREA**
- ▶ Click on the image and select the area with the mouse
- > The control indicates the clicked area with a frame.
- ▶ Shift the area if required by holding down the mouse button

You can fix the drawn area by double-clicking it, thereby protecting it from unintentional shifting.

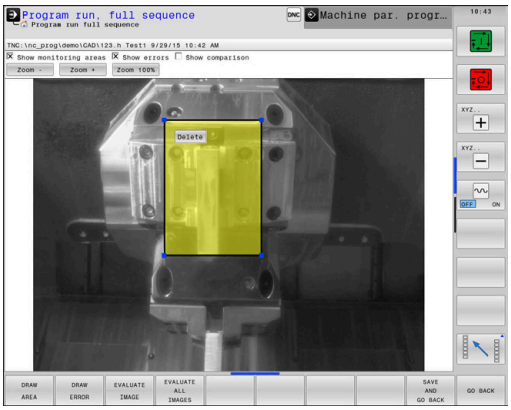
Deleting drawn areas

If you have drawn several monitoring areas or error areas, you can delete these individually.

Proceed as follows:

- ▶ Click on the area you wish to delete
- > The control indicates the clicked area with a frame.
- ▶ Press the **Delete** button

Defining the monitoring area



The monitoring area must be defined in the **Program Run, Full Sequence/Single Block** operating modes. The control will prompt you to define a monitoring area. The control will display this prompt on the screen after you have started the cycle for the first time in the **Program run, Full Sequence/Single Block** operating mode.

A monitoring area consists of one or more windows. If you define multiple windows, they may overlap. The control will only take these areas of the image into account. Errors outside the monitoring area will not be detected. The monitoring area is not linked to the images, but only to the monitoring file specified in **QS600**. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Drawing the monitoring area or error image:

Proceed as follows:

DRAW AREA

- ▶ Press the **DRAW AREA** or **DRAW ERROR** soft key
- ▶ Draw a rectangle around the area to be monitored
- > The control indicates the selected area with a frame.
- ▶ Use the available buttons to resize the image as desired
- ▶ Alternatively, define more windows by pressing the **DRAW AREA** or **DRAW ERROR** soft key and repeat the steps at the desired position
- ▶ Lock the defined area in place by double-clicking.
- > The area is now protected from accidental shifting.

SPEICHERN UND ZURÜCK

- ▶ Press the **SAVE AND GO BACK** soft key
- > The control will save the current image and return to the previous screen.

Deleting drawn areas

Proceed as follows:

- ▶ Select the area to be deleted
- > The control indicates the selected area with a frame.
- ▶ Press the **Delete** button

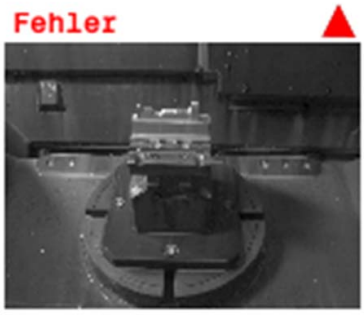
The status display at the top of the image will display information about the minimum number of reference images, the current number of reference images, and the current number of error images.

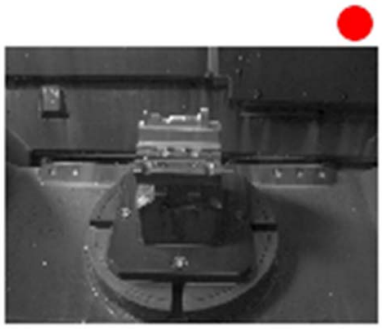
Result of the image evaluation

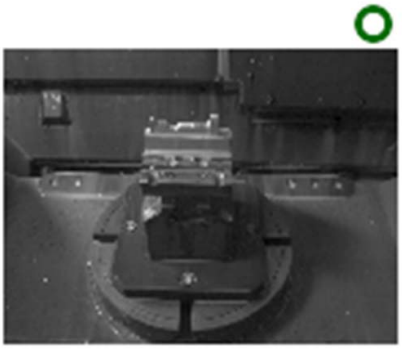
The result of the image evaluation depends on the monitoring area and the reference images. When evaluating all images, each image is evaluated according to the current configuration and the results are compared with the data last saved.

If you change the monitoring area or add/delete reference images, the images may be tagged with the following symbol:

- **Triangle:** You changed the monitoring area or the sensitivity. This has an impact on all of your reference images and the mean image, respectively. As a result of your change to the configuration, the control can no longer detect errors that had been saved for this image! The system has become less sensitive. If you would like to proceed, confirm the reduced sensitivity of the system. The new settings will be applied.
- **Solid circle:** You changed the monitoring area or the sensitivity. This has an impact on all of your reference images and the mean image, respectively. As a result of your change to the configuration, the control can now detect errors that it was unable to detect before. The system has become more sensitive. If you would like to proceed, confirm the increased sensitivity of the system. The new settings will be applied.
- **Empty circle:** No error message: All deviations saved in the image were detected. The system, therefore, has basically kept its previous sensitivity.



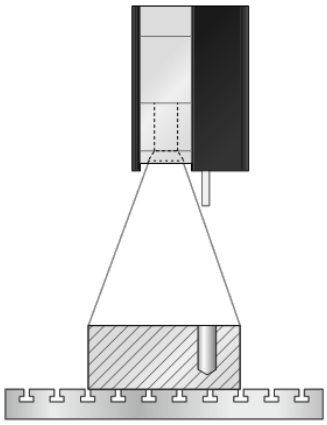




21.2 GLOBAL WORKING SPACE (Cycle 600, DIN/ISO: G600)

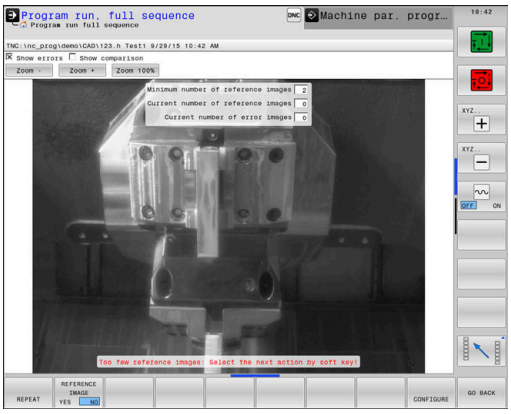
Application

With Cycle 600, GLOBAL WORKING SPACE, you monitor the workspace of your machine tool. The control will generate an image of the current workspace from a position determined by your machine tool builder. Then, the control 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 600 becomes effective as soon as it has been defined, and does not have to be called. Before you can use camera monitoring, you must first generate reference images and define a monitoring area.
Further information: "Creating reference images", Page 791
Further information: "Monitoring phase", Page 792



Creating reference images

Cycle run



1 The machine tool builder mounts the camera onto the main spindle. The main spindle moves to a position defined by the machine tool builder
2 After this position has been reached, the control will automatically open the camera cap
3 Once you run this cycle for the first time in **Program Run, Full Sequence/Single Block**, the control interrupts the current NC program and shows the image taken by the camera.
4 A message is displayed, indicating that there is no reference image to be evaluated
5 Press the **REFERENCE IMAGE YES** soft key
6 Then, the following message is displayed at the bottom of the screen: **Monitoring point not configured: Draw areas!**
7 Press the **CONFIGURE** soft key and define the monitoring area
Further information: "Defining the monitoring area", Page 788
8 These steps are repeated until the control has saved a sufficient number of reference images. The number of reference images can be specified in the cycle with parameter **Q617**
9 Complete the procedure by pressing the **GO BACK** soft key. The control resumes program run
10 Finally, the control closes the camera cap
11 Press **NC Start** and run your NC program as usual

After you have defined the monitoring area, you can press the following soft keys:

GO BACK

- ▶ Press the **GO BACK** soft key
- > The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
Further information: "Result of the image evaluation", Page 789

REPEAT

- ▶ Alternatively, press the **REPEAT** soft key
- > The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
Further information: "Result of the image evaluation", Page 789

REFERENCE IMAGE

- ▶ Alternatively, press the **REFERENCE IMAGE** soft key
- > The status display at the top right displays the word **Reference**. You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the **IMAGE OF ERROR** soft key turns gray.

IMAGE OF ERROR

- Alternatively, press the **ERROR IMAGE** soft key
- > The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an error image can never be a reference image at the same time, the **REFERENCE IMAGES** soft key turns gray.

CONFIGURE

- Alternatively, press the **CONFIGURE** soft key
- > The soft key row changes. You can now change your previously entered settings with respect to the monitoring area and the sensitivity. If you make a change in this menu, this may affect all of your images.
  Further information: "Configuration", Page 787

As soon as the control has generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected, the following message is displayed: **Too few reference images: Select the next action by soft key!** This message disappears once the number of reference images defined in parameter **Q617** has been reached.

The control will generate a mean-value image, taking into account all reference images. During evaluation, new images are compared with the mean-value image, taking the variance into account. Once the specified number of reference images has been reached, the cycle will continue without interruption.

Monitoring phase

Cycle run: Monitoring phase

1 The machine tool builder mounts the camera onto the main spindle. The main spindle moves to a position defined by the machine tool builder
2 After this position has been reached, the control will automatically open the camera cap
3 The control will generate an image of the current condition
4 The control will then compare the images using the mean image and the variance image.
  Further information: "Fundamentals", Page 782
5 Depending on whether the control detects an "error" (deviation), the control can now force a program cancellation. If parameter **Q309**=1 is set, the control will display the image on the screen upon detection of an error. If parameter **Q309**=0 is set, no image will be displayed on the screen and the program will not be aborted.
6 Finally, the control closes the camera cap

Please note while programming:

NOTICE

Danger of collision!

Danger of camera contamination if the camera cap is open (set with parameter **Q613**). This could result in blurred images, or the camera might be damaged.

- Close the camera cap before continuing the machining process

NOTICE

Danger of collision!

Danger of collision with automatic positioning of the camera. The camera and your machine may be damaged.

- Refer to the machine manual to learn about the point the camera is pre-positioned to by the control. Your machine tool builder specifies the coordinates which Cycle 600 positions to.

Your machine must be prepared for camera-based monitoring!

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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:

- Never mark a reference image as an error image at the same time

If you change the monitoring area, this has an impact on all images.

- 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 reasonable number of reference images in parameter **Q617**. (Approx. value: 10 images)
- You can also create more reference images than you have specified in **Q617**

Cycle parameters

- **QS600** (string parameter) **Name of monitoring point?**: Enter the name of your monitoring file.
- **Q616 Feed rate for positioning?**: Feed rate the control uses to position the camera. Here, control moves the camera to a position defined by the machine tool builder.
- **Q309 PGM stop if tolerance exceeded?**: (0/1) Specify whether the control stops the program if an error is detected.
  0: The NC program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter **Q601** is written to even if **Q309**=0.
  1: The NC program will stop after detecting an error, and the generated image will be displayed on the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the control has generated enough reference images. If an error is detected, the control will display a message.
- **Q617 Number of reference images?**: Number of reference images that are required by the control for monitoring.

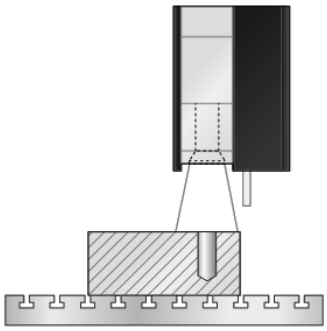
Example

```
4 TCH PROBE 600 GLOBAL WORKING SPACE
   QS600="OS" ;MONITORING POINT
   Q616=500   ;FEED RATE FOR POSITIONING
   Q309=1     ;PGM STOP TOLERANCE
   Q617=10    ;REFERENCE IMAGES
```

21.3 LOCAL WORKING SPACE (Cycle 601)

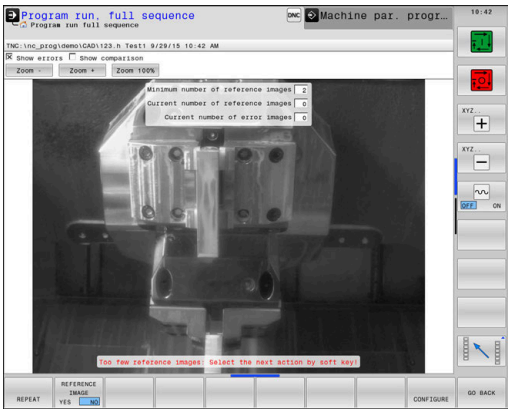
Application

With Cycle 601, Workspace Local, you monitor the workspace of your machine tool. The control will generate an image of the current workspace from the position of the spindle at the moment the cycle is called. Then, the control will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 601 becomes effective as soon as it has been defined, and does not have to be called. Before you can use camera monitoring, you must first generate reference images and define a monitoring area.
Further information: "Creating reference images", Page 795
Further information: "Monitoring phase", Page 797



Creating reference images

Cycle run



1 The machine tool builder mounts the camera onto the main spindle. The main spindle moves to the position programmed previously
2 The control automatically opens the camera cap
3 Once you run this cycle for the first time in **Program Run, Full Sequence/Single Block**, the control interrupts the current NC program and shows the image taken by the camera.
4 A message is displayed, indicating that there is no reference image to be evaluated
5 Press the **REFERENCE IMAGE YES** soft key
6 Then, the following message is displayed at the bottom of the screen: **Monitoring point not configured: Draw areas!**
7 Press the **CONFIGURE** soft key and define the monitoring area
Further information: "Defining the monitoring area", Page 788
8 These steps are repeated until the control has saved a sufficient number of reference images. The number of reference images can be specified in the cycle with parameter **Q617**
9 Complete the procedure by pressing the **GO BACK** soft key. The control resumes program run
10 Finally, the control closes the camera cap
11 Press **NC Start** and run your NC program as usual

After you have defined the monitoring area, you can press the following soft keys:

GO BACK

- Press the **GO BACK** soft key
- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 789)

REPEAT

- Alternatively, press the **REPEAT** soft key
- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 789

REFERENCE IMAGE

- Alternatively, press the **REFERENCE IMAGE** soft key
- The status display at the top right displays the word **Reference**. You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the **IMAGE OF ERROR** soft key turns gray.

IMAGE OF ERROR

- Alternatively, press the **ERROR IMAGE** soft key
- The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an error image can never be a reference image at the same time, the **REFERENCE IMAGES** soft key turns gray.

CONFIGURE

- Alternatively, press the **CONFIGURE** soft key
- The soft key row changes. You can now change your previously entered settings with respect to the monitoring area and the sensitivity. If you make a change in this menu, this may affect all of your images. **Further information:** "Configuration", Page 787)

As soon as the control has generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected, the following message is displayed: **Too few reference images: Select the next action by soft key!** This message disappears once the number of reference images defined in parameter **Q617** has been reached.

The control will generate a mean-value image, taking into account all reference images. During evaluation, new images are compared with the mean-value image, taking the variance into account. Once the specified number of reference images has been reached, the cycle will continue without interruption.

Monitoring phase

The monitoring phase starts as soon as the control has generated enough reference images.

Cycle run: Monitoring phase

1 The machine tool builder mounts the camera onto the main spindle
2 The control automatically opens the camera cap
3 The control will generate an image of the current condition
4 The control will then compare the images using the mean image and the variance image.
5 Depending on whether the control detects an "error" (deviation), the control can now force a program cancellation. If parameter **Q309**=1 is set, the control will display the image on the screen upon detection of an error. If parameter **Q309**=0 is set, no image will be displayed on the screen and the program will not be aborted.
6 Depending on the setting of parameter **Q613**, the control either leaves the camera cap open or closes it.

Please note while programming!

NOTICE

Danger of collision!

Danger of camera contamination if the camera cap is open (set with parameter **Q613**). This could result in blurred images, or the camera might be damaged.

- Close the camera cap before continuing the machining process

Your machine must be prepared for camera-based monitoring!

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

In addition to the Reference Image attribute, you can assign the Error Image attribute to your images. This assignment may impact image evaluation.

Remember the following:

- Never mark a reference image as 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, i.e. 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 reasonable number of reference images in parameter **Q617**. (Approx. value: 10 images)
- You can also create more reference images than you have specified in **Q617**

Cycle parameters

- **QS600** (string parameter) **Name of monitoring point?**: Enter the name of your monitoring file.
- **Q309 PGM stop if tolerance exceeded?**: (0/1) Specify whether the control stops the program if an error is detected.
  0: The NC program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter **Q601** is written to even if **Q309**=0.
  1: The NC program will stop after detecting an error, and the generated image will be displayed on the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the control has generated enough reference images. If an error is detected, the control will display a message.
- **Q613 Keep camera cap open?**: (0/1) Specify whether the control should keep the camera cap open after monitoring.
  0: The control closes the camera cap after running Cycle 601.
  1: The control keeps the camera cap open after running Cycle 601. This function is only recommended if you would like to generate yet another image of the workspace in a different position after the first cycle call of Cycle 601. In order to do so, program the new position in a linear block and then call Cycle 601 with a new monitoring point. Program **Q613**=0 before continuing with the cutting operation!
- **Q617 Number of reference images?**: Number of reference images that are required by the control for monitoring.

Example

```
4 TCH PROBE 601 LOCAL WORKING SPACE
  QS600="OS" ;MONITORING POINT
  Q309=+1    ;PGM STOP TOLERANCE
  Q613=0     ;KEEP CAMERA OPEN
  Q617=10    ;REFERENCE IMAGES
```

21.4 Possible queries

The VSC cycles enter a value in parameter **Q601**.

The following values are possible:

- **Q601** = 1: No error
- **Q601** = 2: Error
- **Q601** = 3: You have yet not defined a monitoring area or you did not save enough reference images
- **Q601** = 10: Internal error (no signal, camera error, etc.)

You can use parameter **Q601** for internal queries.

Further information: If-then decisions: User's Manual Conversational Programming

The following is a programming example for a query:

0 BEGIN PGM 5MM	
1 BLK FORM CYLINDER Z R42 L150	Blank form Cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TCH PROBE 601 LOCAL WORKING SPACE	Define Cycle 600
QS600 = OS ;MONITORING POINT	
Q309=+0 ;PGM STOP TOLERANCE	
Q613 = +0 ;KEEP CAMERA OPEN	
Q617 = 10 ;REFERENCE IMAGES	
4 FN 9: IF Q601 EQU 1 GOTO LBL 20	If parameter Q601 = 1, then jump to LBL 20
5 FN 9: IF Q601 EQU 2 GOTO LBL 21	If parameter Q601 = 2, then jump to LBL 21
6 FN 9: IF Q601 EQU 3 GOTO LBL 22	If parameter Q601 = 3, then jump to LBL 22
7 FN 9: IF Q601 EQU 10 GOTO LBL 23	If parameter Q601 = 10, then jump to LBL 23
8 TOOL CALL "GEAR_HOB_D75"	Call the tool
9 L X+... Y+... R0 FMAX	Program the machining
...	
...	
...	
57 LBL 21	Definition LBL 21
58 STOP	Program stop, the operator can check the condition of the workspace
59 LBL 0	
60 END PGM 5MM	

22

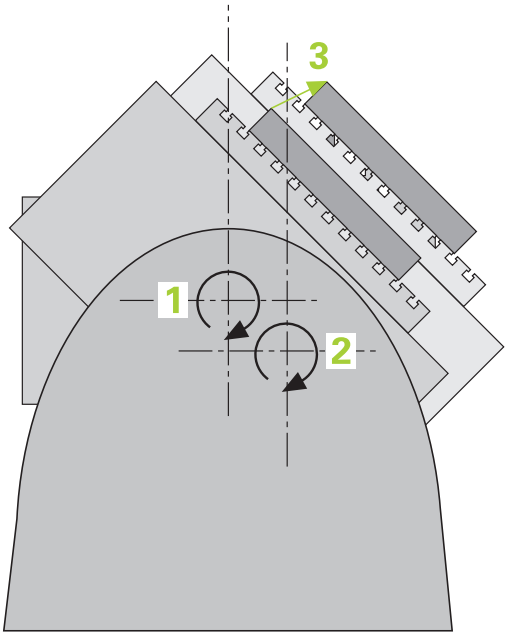
Touch Probe Cycles: Automatic Kinematics Measurement

22.1 Kinematics measurement with TS touch probes (option 48)

Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

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.

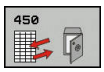


The **KinematicsOpt** function of the control is an important component that helps you to really meet these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached 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 control 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 kinematics table.

Overview

The control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Soft key	Cycle	Page
	450 **SAVE KINEMATICS** Automatic saving and restoring of kinematic configurations	806
	451 **MEASURE KINEMATICS** Automatic checking or optimizing of machine kinematics	809
	452 **PRESET COMPENSATION** Automatic checking or optimizing of machine kinematics	824
	453 **KINEMATICS GRID** Automatic checking or optimizing of machine kinematics	834

22.2 Prerequisites

Refer to your machine manual!
Advanced Function Set 1 (option 8) must have been enabled.
Option 17 must have been enabled.
Option 48 must have been enabled.
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The following are prerequisites for using the KinematicsOpt option:

- 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
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- The machine tool builder must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data.
  - **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value.
  - **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter
  - **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine tool builder and is used to position the rotary axes.

HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

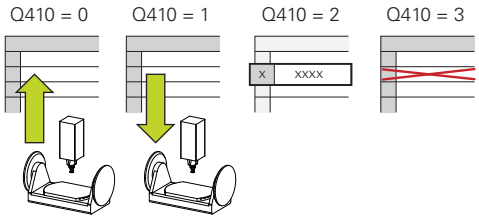
If an M function has been defined in the optional **mStrobeRotAxPos** machine parameter (no. 204803), you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450).

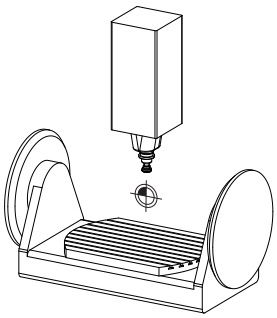
If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

22.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48)

Cycle run

With Touch Probe Cycle 450 you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.





Please note while programming:

Only save and restore data with Cycle 450, while no tool carrier kinematics configuration that includes transformations is active.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

Always save the active kinematics configuration before running a kinematics optimization. Advantage:

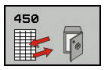
- You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).

With the **Restore** mode, note the following:

- The control can restore saved data only to a matching kinematic configuration
- A change in the kinematics always changes the preset as well. So redefine the preset, if required.

The cycle will not restore identical values, but only data that is different from the existing data. Compensations can only be restored if they had been saved before.

Cycle parameters



- **Q410 Mode (0/1/2/3)?**: Define whether you wish to backup or restore the kinematics configuration:
  0: Backup active kinematics configuration
  1: Restore saved kinematics configuration
  2: Display current memory status
  3: Delete a data record.
- **Q409/QS409 Name of data record?**: Number or name of the record designator. When specifying a number, you can enter values from 0 to 99999, and when entering letters the string length may not exceed 16 characters. 16 memory locations in total are available. **Q409** has no function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all data blocks after confirmation (mode 3). You can use the following wildcards for searching:
  ?: Any single character
  $: Any single alphabetic character (letter)
  #: Any single number
  ***:** Any character string of any length

Saving the current kinematics

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=0       ;MODE
   Q409=947     ;MEMORY DESIGNATION
```

Restoring data blocks

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=1       ;MODE
   Q409=948     ;MEMORY DESIGNATION
```

Displaying all saved data blocks

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=2       ;MODE
   Q409=949     ;MEMORY DESIGNATION
```

Deleting data blocks

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=3       ;MODE
   Q409=950     ;MEMORY DESIGNATION
```

Log function

After running Cycle 450, the control creates a log (**tchpr450.txt**) 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 control 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 control stores the saved data in the file **TNC:\table \DATA450.KD**. This file can be backed up to an external PC with **TNCremo**, for example. If you delete the file, the stored data are removed, too. If the data in the file are changed manually, the data records may become corrupted so that they are unusable.

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).

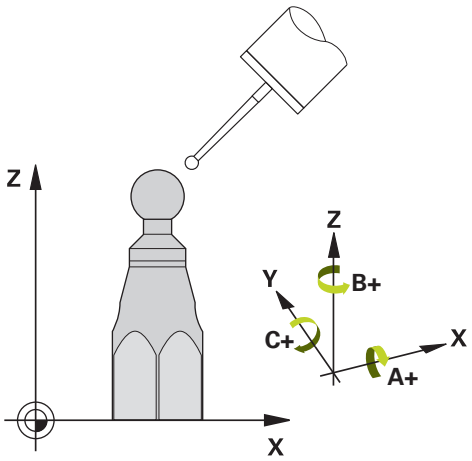
22.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)

Cycle run

Refer to your machine manual!

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 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.



The control will determine 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 Manual Operation mode, set the preset to the center of the sphere or, if you defined **Q431**=1 or **Q431**=3: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select the Program Run operating mode and start the calibration program.
4 The control automatically measures all rotary axes successively in the resolution you defined.
5 The control saves the measured values in the following Q parameters:

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°.

Specify the start and end angles in such a way that the same position is not measured twice. A duplicated point measurement (e.g. measuring positions +90° and -270°) is not advisable, but it will not generate 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

NOTICE

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control 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).

- 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)

NOTICE

Danger of collision!

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine tool builder, enabling the control to move the rotary axes. The machine tool builder must have entered the number of the M function in machine parameter **mStrobeRotAxPos** (no. 244803) for this purpose.

- Note the documentation of the machine tool builder

Define a retraction height greater than 0 if option 2 is not available.

The measured positions are calculated from the start angle, end angle, and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = –30
End angle **Q412** = +90
Number of measuring points **Q414** = 4
Hirth grid = 3°
Calculated stepping angle = (**Q412** – **Q411**) / (**Q414** – 1)
Calculated stepping angle = (90° - (-30°)) / (4 – 1) = 120 / 3 = 40°
Measuring position 1 = **Q411** + 0 * stepping angle = –30° –> –30°
Measuring position 2 = **Q411** + 1 * stepping angle = +10° –> 9°
Measuring position 3 = **Q411** + 2 * stepping angle = +50° –> 51°
Measuring position 4 = **Q411** + 3 * stepping angle = +90° –> 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), e.g. when commissioning the machine.

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 control 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 in such a way on the machine table that with rotary table axes, there is a large measuring circle or that on swivel head axes, 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 performed 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 control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.

Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.

The control does not perform an automatic backlash compensation.

If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can determine the rotary axis backlash (see "Log function", Page 823).

Please note while programming:

If the optional **mStrobeRotAxPos** machine parameter (no. 204803) is not equal to –1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control displays an error message and ends the measurement.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

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 preset to the center of the calibration sphere and activate it, or set input parameter **Q431** to 1 or 3, respectively.

For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.

The control ignores cycle definition data that applies to inactive axes.

For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

A correction in the machine datum (**Q406**=3) is only possible if superimposed rotary axes on the spindle head side or table side are measured.

Angle compensation is only possible with option 52 **KinematicsComp**.

If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

Programming in inches: The control always records the log data and results of measurement in millimeters.

During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Cycle parameters

- **Q406 Mode (0/1/2/3)?**: Specify whether the control should check or optimize the active kinematics:
  0: Check active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The control displays the results of measurement in a measurement log.
  1: Optimize active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **the position of the rotary axes** of the active kinematics.
  2: Optimize active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **angle and positioning errors.** Software option 52, KinematicsComp, is required for the compensation of angle errors.
  3: Optimize active machine kinematics: The control automatically corrects the machine datum. It then optimizes **angle and positioning errors.** Software option 52, KinematicsComp, is required.
- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999Alternatively **PREDEF**
- **Q408 Retraction height?** (absolute): Input range: 0.0001 to 99999.9999
  0: Do not move to retraction height. The control moves the touch probe to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C
  >0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**.
- **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q380 Ref. angle in ref. axis?** (absolute): Enter the 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

Saving and checking the kinematics

```
4 TOOL CALL  "TCH PROBE" Z
5 TCH PROBE 450  SAVE KINEMATICS
    Q410=0      ;MODE
    Q409=5      ;MEMORY DESIGNATION
6 TCH PROBE 451  MEASURE
  KINEMATICS
    Q406=0      ;MODE
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=750    ;F PRE-POSITIONING
    Q380=0      ;REFERENCE ANGLE
    Q411=-90    ;START ANGLE A AXIS
    Q412=+90    ;END ANGLE A AXIS
    Q413=0      ;INCID. ANGLE A AXIS
    Q414=0      ;MEAS. POINTS A AXIS
    Q415=-90    ;START ANGLE B AXIS
    Q416=+90    ;END ANGLE B AXIS
    Q417=0      ;INCID. ANGLE B AXIS
    Q418=2      ;MEAS. POINTS B AXIS
    Q419=-90    ;START ANGLE C AXIS
    Q420=+90    ;END ANGLE C AXIS
    Q421=0      ;INCID. ANGLE C AXIS
    Q422=2      ;MEAS. POINTS C AXIS
    Q423=4      ;NO. OF PROBE POINTS
    Q431=0      ;PRESET
    Q432=0      ;BACKLASH, ANG. RANGE
```

- **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range: –359.999 to 359.999
- **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range: –359.999 to 359.999
- **Q413 Angle of incidence in A axis?**: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range: –359.999 to 359.999
- **Q414 No. of meas. points in A (0...12)?**: Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12
- **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range: –359.999 to 359.999
- **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range: –359.999 to 359.999
- **Q417 Angle of incidence in B axis?**: Inclination angle in the B axis at which the other rotary axes are to be measured. Input range: –359.999 to 359.999
- **Q418 No. of meas. points in B (0...12)?**: Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12
- **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range: –359.999 to 359.999
- **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range: –359.999 to 359.999
- **Q421 Angle of incidence in C axis?**: Inclination angle in the C axis at which the other rotary axes are to be measured. Input range: –359.999 to 359.999

- **Q422 No. of meas. points in C (0...12)?**: Number of probe measurements to be used for measuring the C axis. Input range: 0 to 12. If the input value = 0, the control does not measure the respective axis.
- **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Fewer measuring points increase speed and more measuring points increase measurement precision.
- **Q431 Preset (0/1/2/3)?** Define whether the control automatically sets the active preset to the center of the sphere:
  0: Do not define the preset automatically at the center of the sphere: Preset manually before cycle start
  1: Define the preset automatically at the center of the sphere (this overwrites the active preset): Manually pre-position the touch probe before cycle start using the calibration sphere
  2: Automatically define the preset at the center of the sphere after measuring): Preset manually before cycle start
  3: Define the preset before and after measurement at the center of the sphere (the active preset will be overwritten): Pre-position the touch probe manually before cycle start using the calibration sphere
- **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input range: –3.0000 to +3.0000

If you have activated presetting before the calibration (**Q431** = 1/3), then move the touch probe to the set-up 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 control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

"Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data are adjusted automatically.

Position and Angle Optimization mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

```
1 TOOL CALL  “TCH PROBE“ Z
2 TCH PROBE 451  MEASURE
   KINEMATICS
    Q406=1       ;MODE
    Q407=12.5    ;SPHERE RADIUS
    Q320=0       ;SET-UP CLEARANCE
    Q408=0       ;RETR. HEIGHT
    Q253=750     ;F PRE-POSITIONING
    Q380=0       ;REFERENCE ANGLE
    Q411=-90     ;START ANGLE A AXIS
    Q412=+90     ;END ANGLE A AXIS
    Q413=0       ;INCID. ANGLE A AXIS
    Q414=0       ;MEAS. POINTS A AXIS
    Q415=-90     ;START ANGLE B AXIS
    Q416=+90     ;END ANGLE B AXIS
    Q417=0       ;INCID. ANGLE B AXIS
    Q418=0       ;MEAS. POINTS B AXIS
    Q419=+90     ;START ANGLE C AXIS
    Q420=+270    ;END ANGLE C AXIS
    Q421=0       ;INCID. ANGLE C AXIS
    Q422=3       ;MEAS. POINTS C AXIS
    Q423=3       ;NO. OF PROBE POINTS
    Q431=1       ;PRESET
    Q432=0.5     ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle 451, the control will create a log **(TCHPR451.html)** and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

22.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48)

Cycle run

Refer to your machine manual!

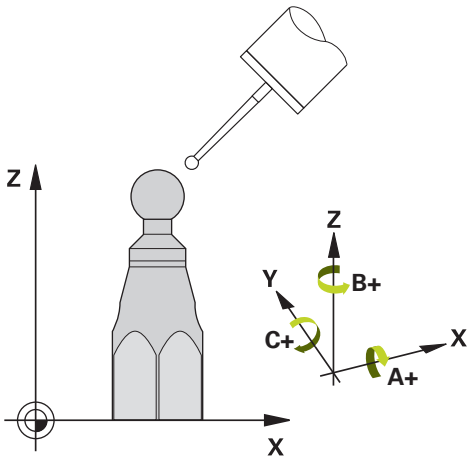
Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 809). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the 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 Set the preset in the calibration sphere.
3 Set the preset on the workpiece, and start machining the workpiece.
4 Use Cycle 452 for preset compensation at regular intervals. The control measures the drift of the axes involved and compensates it in the kinematics description.



Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (–1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (–1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Please note while programming:

If the kinematics data determined exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control displays an error message and ends the measurement.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

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.

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 preset 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 an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.

For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

Programming in inches: The control always records the log data and results of measurement in millimeters.

Cycle parameters

- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q408 Retraction height?** (absolute): Input range: 0.0001 to 99999.9999
  0: Do not move to retraction height. The control moves the touch probe to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C
  >0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**.
- **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q380 Ref. angle in ref. axis?** (absolute): Enter the reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000
- **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range: –359.999 to 359.999
- **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range: –359.999 to 359.999
- **Q413 Angle of incidence in A axis?**: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range: –359.999 to 359.999
- **Q414 No. of meas. points in A (0...12)?**: Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12

Calibration program

```
4 TOOL CALL  “TCH PROBE“ Z
5 TCH PROBE 450  SAVE KINEMATICS
    Q410=0      ;MODE
    Q409=5      ;MEMORY DESIGNATION
6 TCH PROBE 452  PRESET
  COMPENSATION
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=750    ;F PRE-POSITIONING
    Q380=0      ;REFERENCE ANGLE
    Q411=-90    ;START ANGLE A AXIS
    Q412=+90    ;END ANGLE A AXIS
    Q413=0      ;INCID. ANGLE A AXIS
    Q414=0      ;MEAS. POINTS A AXIS
    Q415=-90    ;START ANGLE B AXIS
    Q416=+90    ;END ANGLE B AXIS
    Q417=0      ;INCID. ANGLE B AXIS
    Q418=2      ;MEAS. POINTS B AXIS
    Q419=-90    ;START ANGLE C AXIS
    Q420=+90    ;END ANGLE C AXIS
    Q421=0      ;INCID. ANGLE C AXIS
    Q422=2      ;MEAS. POINTS C AXIS
    Q423=4      ;NO. OF PROBE POINTS
    Q432=0      ;BACKLASH, ANG. RANGE
```

- **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range: –359.999 to 359.999
- **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range: –359.999 to 359.999
- **Q417 Angle of incidence in B axis?**: Inclination angle in the B axis at which the other rotary axes are to be measured. Input range: –359.999 to 359.999
- **Q418 No. of meas. points in B (0...12)?**: Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12
- **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range: –359.999 to 359.999
- **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range: –359.999 to 359.999
- **Q421 Angle of incidence in C axis?**: Inclination angle in the C axis at which the other rotary axes are to be measured. Input range: –359.999 to 359.999
- **Q422 No. of meas. points in C (0...12)?**: Number of probe measurements to be used for measuring the C axis. Input range: 0 to 12. If the input value = 0, the control does not measure the respective axis.
- **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Fewer measuring points increase speed and more measuring points increase measurement precision.
- **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input range: –3.0000 to +3.0000

Adjustment of interchangeable heads

The goal of this procedure is to achieve that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, the adjustment of a fork head with A and C axes is described. 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.
- Define the preset (using **Q431** = 2 or 3 in Cycle 451) after measuring the reference head

Measuring a reference head

```
1 TOOL CALL  “TCH PROBE“ Z
2 TCH PROBE  451 MEASURE
   KINEMATICS
     Q406=1       ;MODE
     Q407=12.5    ;SPHERE RADIUS
     Q320=0       ;SET-UP CLEARANCE
     Q408=0       ;RETR. HEIGHT
     Q253=2000    ;F PRE-POSITIONING
     Q380=+45     ;REFERENCE ANGLE
     Q411=-90     ;START ANGLE A AXIS
     Q412=+90     ;END ANGLE A AXIS
     Q413=45      ;INCID. ANGLE A AXIS
     Q414=4       ;MEAS. POINTS A AXIS
     Q415=-90     ;START ANGLE B AXIS
     Q416=+90     ;END ANGLE B AXIS
     Q417=0       ;INCID. ANGLE B AXIS
     Q418=2       ;MEAS. POINTS B AXIS
     Q419=+90     ;START ANGLE C AXIS
     Q420=+270    ;END ANGLE C AXIS
     Q421=0       ;INCID. ANGLE C AXIS
     Q422=3       ;MEAS. POINTS C AXIS
     Q423=4       ;NO. OF PROBE POINTS
     Q431=3       ;PRESET
     Q432=0       ;BACKLASH, ANG. RANGE
```

- Load 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 entire process.
- All other interchangeable heads can be adjusted in the same way

The head change function can vary depending on the individual machine tool. Refer to your machine manual.

Adjusting an interchangeable head

```
3 TOOL CALL  "TCH PROBE" Z
4 TCH PROBE  452 PRESET
  COMPENSATION
    Q407=12.5  ;SPHERE RADIUS
    Q320=0     ;SET-UP CLEARANCE
    Q408=0     ;RETR. HEIGHT
    Q253=2000  ;F PRE-POSITIONING
    Q380=+45   ;REFERENCE ANGLE
    Q411=-90   ;START ANGLE A AXIS
    Q412=+90   ;END ANGLE A AXIS
    Q413=45    ;INCID. ANGLE A AXIS
    Q414=4     ;MEAS. POINTS A AXIS
    Q415=-90   ;START ANGLE B AXIS
    Q416=+90   ;END ANGLE B AXIS
    Q417=0     ;INCID. ANGLE B AXIS
    Q418=2     ;MEAS. POINTS B AXIS
    Q419=+90   ;START ANGLE C AXIS
    Q420=+270  ;END ANGLE C AXIS
    Q421=0     ;INCID. ANGLE C AXIS
    Q422=0     ;MEAS. POINTS C AXIS
    Q423=4     ;NO. OF PROBE POINTS
    Q432=0     ;BACKLASH, ANG. RANGE
```

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
- Define the preset (using **Q432** = 2 or 3 in Cycle 451) after measuring the kinematics
- Then set the presets on your workpiece and start the machining process.

Reference measurement for drift compensation

```
1 TOOL CALL “TCH PROBE“ Z
2 CYCL DEF 247 PRESETTING
    Q339=1       ;PRESET NUMBER
3 TCH PROBE  451 MEASURE
  KINEMATICS
    Q406=1       ;MODE
    Q407=12.5    ;SPHERE RADIUS
    Q320=0       ;SET-UP CLEARANCE
    Q408=0       ;RETR. HEIGHT
    Q253=750     ;F PRE-POSITIONING
    Q380=+45     ;REFERENCE ANGLE
    Q411=+90     ;START ANGLE A AXIS
    Q412=+270    ;END ANGLE A AXIS
    Q413=45      ;INCID. ANGLE A AXIS
    Q414=4       ;MEAS. POINTS A AXIS
    Q415=-90     ;START ANGLE B AXIS
    Q416=+90     ;END ANGLE B AXIS
    Q417=0       ;INCID. ANGLE B AXIS
    Q418=2       ;MEAS. POINTS B AXIS
    Q419=+90     ;START ANGLE C AXIS
    Q420=+270    ;END ANGLE C AXIS
    Q421=0       ;INCID. ANGLE C AXIS
    Q422=3       ;MEAS. POINTS C AXIS
    Q423=4       ;NO. OF PROBE POINTS
    Q431=3       ;PRESET
    Q432=0       ;BACKLASH, ANG. RANGE
```

- Measure the drift of the axes at regular intervals.
- Insert the touch probe
- Activate the preset in the calibration sphere.
- Use Cycle 452 to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the entire process.

This procedure can also be performed on machines without rotary axes.

Drift compensation

```
4 TOOL CALL “TCH PROBE“ Z
5 TCH PROBE  452 PRESET
  COMPENSATION
    Q407=12.5  ;SPHERE RADIUS
    Q320=0     ;SET-UP CLEARANCE
    Q408=0     ;RETR. HEIGHT
    Q253=99999 ;F PRE-POSITIONING
    Q380=+45   ;REFERENCE ANGLE
    Q411=-90   ;START ANGLE A AXIS
    Q412=+90   ;END ANGLE A AXIS
    Q413=45    ;INCID. ANGLE A AXIS
    Q414=4     ;MEAS. POINTS A AXIS
    Q415=-90   ;START ANGLE B AXIS
    Q416=+90   ;END ANGLE B AXIS
    Q417=0     ;INCID. ANGLE B AXIS
    Q418=2     ;MEAS. POINTS B AXIS
    Q419=+90   ;START ANGLE C AXIS
    Q420=+270  ;END ANGLE C AXIS
    Q421=0     ;INCID. ANGLE C AXIS
    Q422=3     ;MEAS. POINTS C AXIS
    Q423=3     ;NO. OF PROBE POINTS
    Q432=0     ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle 452, the control creates a log (**TCHPR452.html**) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes
  - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

Notes on log data

(see "Log function", Page 823)

22.6 KINEMATICS GRID (Cycle 453, DIN/ISO: G453, option 48)

Cycle run

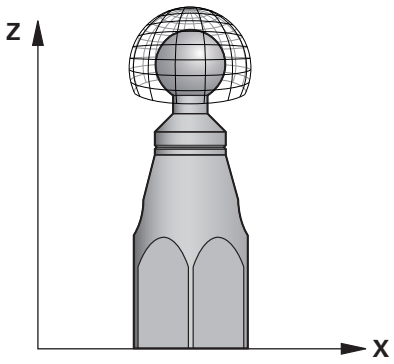
Refer to your machine manual!

KinematicsOpt (software option 48) is required.

KinematicsComp (software option 52) is required.

This function must be enabled and adapted by the machine tool builder.

To use this cycle, your machine tool builder needs to create and configure a compensation table (*.kco) first and perform some more settings.



Even if your machine was already optimized regarding positioning errors (e.g. via Cycle 451), residual errors at the Tool Center Point (**TCP**) during tilting of the rotary axes may remain. Such errors occur particularly with swivel-head machines. These can result, for example, from component errors (e.g. a bearing error) with head rotation axes.

Cycle 453 **KINEMATICS GRID** enables these errors to be detected and compensated in accordance with the tilting axis positions. Options 48 (**KinematicsOpt**) and 52 (**KinematicsComp**) are required. With this cycle and using a 3-D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine tool builder defines these tilting axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing process on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine tool builder defines this compensation table (*.kco) and specifies its storage location.

When using Cycle 453, run it at different positions in the workspace. This allows you to check immediately if a compensation with Cycle 453 has the desired positive effect on the machine's accuracy. Only when the desired improvements are achieved with the same compensation values at several positions is such a type of compensation suitable for the respective machine. If this is not the case, then the errors are to be sought outside the rotary axes.

Perform the measurement with Cycle 453 in an optimized condition regarding the rotary axis positioning errors. For this purpose, use e.g. Cycle 451 beforehand.

HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting from a measurement in a compensation table (*.kco). (This applies to mode **Q406**=1.)

1 Clamp the calibration sphere and check for potential collisions.
2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select one of the Program Run operating modes and start the NC program
4 The cycle is executed in accordance with the setting in **Q406** (–1=Delete mode / 0=Test mode / 1=Compensate mode)

Various modes (Q406)

Delete mode Q406 = -1

- The axes are not moved
- The control writes all values to the compensation table (*.kco), setting them to "0".The result is that no further compensations will be effective for the currently selected kinematics.

Test mode Q406 = 0

- The control probes the calibration sphere.
- The results are saved to a log in html format that is stored in the directory where the current NC program resides.

Compensate mode Q406 = 1

- The control probes the calibration sphere.
- The control writes the deviations to the compensation table (*.kco). The table is updated and the compensation settings are immediately effective.
- The results are saved to a log in html format that is stored in the directory where the current NC program resides.

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. It is recommended to clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

Please note while programming:

KinematicsOpt (software option 48) is required. KinematicsComp (software option 52) is required.

This function must be enabled and adapted by the machine tool builder.

The machine tool builder defines the storage location of the compensation table (*.kco).

If the optional **mStrobeRotAxPos** machine parameter (no. 204803) is not equal to –1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.

When probing, the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control repeats the measurement and only then displays an error message and ends the measurement.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

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 preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.

For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.

Programming in inches: The control always records the log data and results of measurement in millimeters.

During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.

Cycle parameters

- **Q406 Mode (-1/0/+1)** : Define whether the control should write the values to the compensation table (*.kco) with 0, check the currently existing deviations, or perform a compensation. A log (*.html) is created.
  -1: Delete values in the compensation table (*.kco). The compensation values for TCP positioning errors are set to 0 in the compensation table (*.kco). The control will not perform any probing. No results will be output to the log (*.html).
  0: Check TCP positioning errors. The control measures the TCP positioning errors in accordance with the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html).
  1: Compensate TCP positioning errors. The control measures the TCP positioning errors in accordance with the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html).
- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999Alternatively **PREDEF**
- **Q408 Retraction height?** (absolute): Input range: 0.0001 to 99999.9999
  0: Do not move to retraction height. The control moves the touch probe to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C
  >0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**.

Probing with Cycle 453

```
4 TOOL CALL "TCH PROBE" Z
6 TCH PROBE 453 KINEMATICS GRID
   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
   Q423=4      ;NO. OF PROBE POINTS
   Q431=0      ;PRESET
```

- **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q380 Ref. angle in ref. axis?** (absolute): Enter the 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
- **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Fewer measuring points increase speed and more measuring points increase measurement precision.
- **Q431 Preset (0/1/2/3)?** Define whether the control automatically sets the active preset to the center of the sphere:
  0: Do not define the preset automatically at the center of the sphere: Preset manually before cycle start
  1: Define the preset automatically at the center of the sphere (this overwrites the active preset): Manually pre-position the touch probe before cycle start using the calibration sphere
  2: Automatically define the preset at the center of the sphere after measuring): Preset manually before cycle start
  3: Define the preset before and after measurement at the center of the sphere (the active preset will be overwritten): Pre-position the touch probe manually before cycle start using the calibration sphere

If you have activated presetting before the calibration (**Q431** = 1/3), then move the touch probe to the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.

Log function

After running Cycle 453, the control will create a log (**TCHPR453.html**) and save it in the folder where the current NC program resides. It contains the following data:

- Date and time of protocol creation
- Path of the NC program from which the cycle was run
- Number and name of the currently active tool
- Mode
- Measured data: Standard deviation and maximum deviation
- Information at which position in degrees (°) the maximum deviation occurred
- Number of measuring positions

23

Touch Probe Cycles: Automatic Tool Measurement

23.1 Fundamentals

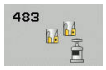
Overview

Refer to your machine manual!
Some cycles and functions may not be provided on your machine.
Option 17 is required.
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Operating notes
- When running touch probe cycles, Cycle **8 MIRROR IMAGE**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active
- HEIDENHAIN only guarantees the proper operation of the probing cycles if HEIDENHAIN touch probes are used.

In conjunction with the control'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.T tool file and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:
- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

You can program the cycles for tool measurement in the **Programming** mode of operation using the **TOUCH PROBE** key. The following cycles are available:

New format	Old format	Cycle	Page
		Calibrating the TT, Cycles 30 and 480	848
		Measuring the tool length, Cycles 31 and 481	850
		Measuring the tool radius, Cycles 32 and 482	854
		Measuring the tool length and radius, Cycles 33 and 483	858
		Calibrating the wireless TT 449, Cycle 484	862

The measuring cycles can be used only when the central tool file TOOL.T is active.

Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in controls for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter **Q199**.

Setting machine parameters

The tool touch probe cycles 480, 481, 482, 483, and 484 can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).

Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200).

When measuring a stationary tool, the Control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius [mm]

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

v:	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (no. 122710) = **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 lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance (**measureTolerance1** (no. 122715), the sooner you will encounter this effect.

probingFeedCalc (no. 122710) = **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 control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = (r • **measureTolerance1**)/ 5 mm) where

r:	Active tool radius [mm]
measureTolerance1:	Maximum permissible error of measurement

Entries in the TOOL.T tool table

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**Cutting direction (M3 = -)?**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: 4 cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: 4 cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

23.2 Calibrating the TT (Cycle 30 or 480, DIN/ISO: G480)

Cycle run

Refer to your machine manual!

The TT is calibrated with measuring cycle TCH PROBE 30 or TCH PROBE 480. (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 843). The calibration process runs automatically. The control 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 cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Calibration process:

1 Clamp the calibrating tool. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin
2 Manually position the calibrating tool in the working plane via the center of the TT
3 Position the calibrating tool in the tool axis approx. 15 mm + safety clearance above the TT
4 The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
5 The calibration process along the tool axis starts
6 Calibration then follows in the working plane
7 The control positions the calibrating tool in the working plane at a position of TT radius + set-up clearance + 11 mm
8 Then the TNC moves the tool downwards along the tool axis and the calibration process starts
9 During probing, the control moves in a square pattern
10 The control saves the calibration values and considers them during subsequent tool measurement
11 The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Please note while programming:

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The functioning of the calibration cycle is dependent on machine parameter **CfgTTRoundStylus** (No. 114200). Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the TOOL.T tool table.

The position of the TT within the machine working space must be defined by setting machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**.

If you change the setting of any of the machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**, you must recalibrate.

Cycle parameters

- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)). Input range: –99999.9999 to 99999.9999

Example of old format

```
6 TOOL CALL  1 Z
7 TCH PROBE 30.0  CALIBRATE TT
8 TCH PROBE 30.1  HEIGHT: +90
```

Example of new format

```
6 TOOL CALL  1 Z
7 TCH PROBE 480  CALIBRATE TT
   Q260=+100 ;CLEARANCE HEIGHT
```

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

Cycle run

Refer to your machine manual!

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 481 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483"). Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.
- 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 stationary.

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 (**R-OFFS**).

Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control 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** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating 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 in Cycle TCH PROBE 31, set parameter Probe the teeth = 1.

Please note while programming:

NOTICE

Danger of collision!

To evaluate **Q199**, you need to set **stopOnCheck** (no. 122717) to **FALSE**. The NC program is not stopped when the breakage tolerance is exceeded. There is a danger of collision!

- Make sure that the NC program stops automatically if the breakage tolerance is exceeded.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: 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

- **Q340 Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for parameter **Q115**. If the delta value is greater than the permissible tool length tolerance for wear or breakage, the control will lock the tool (status L in TOOL.T)
  2: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q115**. Nothing is entered under L or DL in the tool table.
- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: –99999.9999 to 99999.9999
- **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- **For further information**, Page 853

Example of new format

```
6 TOOL CALL  12 Z
7 TCH PROBE 481  CAL. TOOL LENGTH
   Q340=1       ;CHECK
   Q260=+100 ;CLEARANCE HEIGHT
   Q341=1       ;PROBING THE TEETH
```

Cycle 31 includes an additional parameter:

- **Parameter number for result?**: Parameter number in which the control saves the status of measurement results:
  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 in your NC program, answer the dialog prompt with **NO ENT**.

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 31.0  CAL. TOOL LENGTH
8 TCH PROBE 31.1  CHECK: 0
9 TCH PROBE 31.2  HEIGHT: +120
10 TCH PROBE 31.3  PROBING THE
    TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 31.0  CAL. TOOL LENGTH
8 TCH PROBE 31.1  CHECK: 1  Q5
9 TCH PROBE 31.2  HEIGHT: +120
10 TCH PROBE 31.3  PROBING THE
    TEETH: 1
```

23.4 Measuring a tool radius (Cycle 32 or 482, ISO: G482)

Cycle run

Refer to your machine manual!

To measure a 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 843). Select via input parameters by which of two methods the tool radius 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 control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

NOTICE

Danger of collision!

To evaluate **Q199**, you need to set **stopOnCheck** (no. 122717) to **FALSE**. The NC program is not stopped when the breakage tolerance is exceeded. There is a danger of collision!

- Make sure that the NC program stops automatically if the breakage tolerance is exceeded.

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine **CfgTT** (no. 122700). Refer to your machine manual.

Cycle parameters

- **Q340 Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for parameter **Q116**. If the delta value is greater than the permissible tool radius tolerance for wear or breakage, the control will lock the tool (status L in TOOL.T)
  2: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q116**. Nothing is entered under R or DR in the tool table.
- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: –99999.9999 to 99999.9999
- **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- **For further information**, Page 857

Example of new format

```
6 TOOL CALL  12 Z
7 TCH PROBE 482  CAL. TOOL RADIUS
   Q340=1       ;CHECK
   Q260=+100 ;CLEARANCE HEIGHT
   Q341=1       ;PROBING THE TEETH
```

Cycle 32 includes an additional parameter:

- **Parameter number for result?**: Parameter number in which the control saves the status of the measurement results:
  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 in your NC program, answer the dialog prompt with **NO ENT**.

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 32.0  CAL. TOOL RADIUS
8 TCH PROBE 32.1  CHECK: 0
9 TCH PROBE 32.2  HEIGHT: +120
10 TCH PROBE 32.3  PROBING THE
    TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 32.0  CAL. TOOL RADIUS
8 TCH PROBE 32.1  CHECK: 1  Q5
9 TCH PROBE 32.2  HEIGHT: +120
10 TCH PROBE 32.3  PROBING THE
    TEETH: 1
```

23.5 Measuring tool length and radius (Cycle 33 or 483, ISO: G483)

Cycle run

Refer to your machine manual!

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 843). 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 control 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 481 and 482.

Please note while programming:

NOTICE

Danger of collision!

To evaluate **Q199**, you need to set **stopOnCheck** (no. 122717) to **FALSE**. The NC program is not stopped when the breakage tolerance is exceeded. There is a danger of collision!

- Make sure that the NC program stops automatically if the breakage tolerance is exceeded.

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine parameter **CfgTT** (no. 122700). Refer to your machine manual.

Cycle parameters

- **Q340 Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL=0 and DR=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in Q parameters **Q115** and **Q116**. If the delta value is greater than the permissible tool length or radius tolerance for wear or breakage, the control will lock the tool (status L in TOOL.T)
  2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The control calculates the deviation from the stored values and writes it to Q parameter **Q115** or **Q116**. Nothing is entered under L, R, or DL, DR in the tool table.
- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: –99999.9999 to 99999.9999
- **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- **For further information**, Page 861

Example of new format

```
6 TOOL CALL  12 Z
7 TCH PROBE 483  MEASURE TOOL
   Q340=1      ;CHECK
   Q260=+100 ;CLEARANCE HEIGHT
   Q341=1      ;PROBING THE TEETH
```

Cycle 33 includes an additional parameter:

- **Parameter number for result?**: Parameter number in which the control saves the status of the measurement results:
  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 in your NC program, answer the dialog prompt with **NO ENT**.

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 33.0  MEASURE TOOL
8 TCH PROBE 33.1  CHECK: 0
9 TCH PROBE 33.2  HEIGHT: +120
10 TCH PROBE 33.3  PROBING THE
    TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 33.0  MEASURE TOOL
8 TCH PROBE 33.1  CHECK: 1  Q5
9 TCH PROBE 33.2  HEIGHT: +120
10 TCH PROBE 33.3  PROBING THE
    TEETH: 1
```

23.6 Calibrating the wireless TT 449 (Cycle 484, 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

Refer to your machine manual!

To calibrate the tool touch probe, program measuring cycle TCH PROBE 484. In 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 control interrupts the calibration cycle and displays a dialog in a new window.
- ▶ You are prompted 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 run without stop. 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 cylindrical pin. Enter the exact length and radius of the calibrating tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

NOTICE

Danger of collision!

To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536**=1! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibrating tool whose diameter is too small and/or that protrudes too far from the chuck.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

- **Q536 Stop before running (0=Stop)?**: Specify whether to stop before cycle start or run the cycle automatically without stopping:
  0: Stop before running the cycle. You are prompted in a dialog to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press **NC Start** to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process
  1: No stop before running the cycle. The control starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

Example

```
6 TOOL CALL  1 Z
7 TCH PROBE 484  CALIBRATE TT
    Q536=+0     ;STOP BEFORE RUNNING
```

24

Tables of Cycles

24.1 Table of cycles

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active	Page
7	Datum shift	■		219
8	Mirroring	■		226
9	Dwell time	■		357
10	Rotation	■		228
11	Scaling factor	■		230
12	Program call	■		358
13	Oriented spindle stop	■		359
14	Contour definition	■		261
18	Thread cutting		■	413
19	Tilting the working plane	■		233
20	Contour data SL II	■		266
21	Pilot drilling SL II		■	268
22	Rough out SL II		■	270
23	Floor finishing SL II		■	275
24	Side finishing SL II		■	277
25	Contour train		■	281
26	Axis-specific scaling	■		231
27	Cylinder surface		■	323
28	Cylindrical surface slot		■	326
29	Cylinder surface ridge		■	330
32	Tolerance	■		360
39	Cylinder surface contour		■	333
200	Drilling		■	85
201	Reaming		■	87
202	Boring		■	89
203	Universal drilling		■	92
204	Back boring		■	98
205	Universal pecking		■	102
206	Tapping with a floating tap holder, new		■	129
207	Rigid tapping, new		■	132
208	Bore milling		■	110
209	Tapping with chip breaking		■	136
220	Polar pattern	■		246
221	Cartesian pattern	■		249

Cycle number	Cycle name	DEF active	CALL active	Page
224	Datamatrix code pattern	■		251
225	Engraving		■	382
232	Face milling		■	388
233	Face milling (selectable milling direction, consider the side walls)		■	206
238	Measure machine status	■		409
239	Ascertain the load	■		411
240	Centering		■	121
241	Single-lip deep-hole drilling		■	113
247	Presetting	■		240
251	Rectangular pocket (complete machining)		■	167
252	Circular pocket (complete machining)		■	173
253	Slot milling		■	180
254	Circular slot		■	185
256	Rectangular stud (complete machining)		■	191
257	Circular stud (complete machining)		■	196
258	Polygon stud		■	200
262	Thread milling		■	143
263	Thread milling/countersinking		■	147
264	Thread drilling/milling		■	151
265	Helical thread drilling/milling		■	155
267	Outside thread milling		■	159
270	Contour train data		■	280
271	OCM contour data		■	307
272	OCM roughing		■	309
273	OCM finishing floor		■	312
274	OCM finishing side		■	314
275	Trochoidal slot		■	285
276	3-D contour train		■	291
285	Define gear	■		395
286	Gear hobbing		■	398
287	Gear skiving		■	404
291	Coupling turning interpolation		■	371
292	Contour turning interpolation		■	364

Turning cycles

Cycle number	Cycle name	DEF active	CALL active	Page
800	Adapt rotary coordinate system	■		432
801	Reset the turning system	■		440
810	Turn contour, longitudinal		■	466
811	Turn shoulder, longitudinal		■	454
812	Turn shoulder, longitudinal extended		■	457
813	Turn, longitudinal plunge		■	460
814	Turn, longitudinal plunge extended		■	463
815	Turn contour-parallel		■	470
820	Turn contour, transverse		■	487
821	Turn shoulder face		■	473
822	Turn shoulder face extended		■	476
823	Turn, transverse plunge		■	480
824	Turn, transverse plunge extended		■	483
830	Thread, contour-parallel		■	544
831	Thread, longitudinal		■	536
832	Thread, extended		■	540
840	Radial contour recessing		■	505
841	Simple radial recessing		■	491
842	Extended radial recessing		■	494
850	Axial contour recessing		■	509
851	Simple axial recessing		■	498
852	Extended axial recessing		■	501
860	Recessing contour, radial		■	528
861	Recessing, radial		■	513
862	Recessing, radial extended		■	516
870	Recessing contour, axial		■	532
871	Recessing, axial		■	520
872	Recessing, axial extended		■	523
880	Gear hobbing		■	442
883	Turning, simultaneous finishing		■	548
892	Check unbalance	■		450

Grinding cycles

Cycle number	Cycle name	DEF active	CALL active	Page
1000	Define reciprocating stroke	■		565
1001	Start reciprocating stroke	■		568
1002	Stop reciprocating stroke	■		569
1010	Dressing diameter	■		570
1015	Profile dressing	■		574
1030	Activate wheel edge	■		578
1032	Grinding wheel length compensation	■		580
1033	Grinding wheel radius compensation	■		582

Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	Reference plane	■		712
1	Polar preset	■		713
3	Measuring	■		755
4	Measuring in 3-D	■		757
444	Probing in 3D	■		759
30	Calibrating a TT	■		848
31	Measure/Inspect the tool length	■		850
32	Measure/Inspect the tool radius	■		854
33	Measure/Inspect the tool length and the tool radius	■		858
400	Basic rotation: Rot of 2 points	■		623
401	Basic rotation: Rot. of 2 holes	■		626
402	Basic rotation: Rot. of 2 studs	■		630
403	Compensate misalignment with rotary axis	■		635
404	Set basic rotation	■		644
405	Compensate misalignment with the C axis	■		640
408	Slot center preset (FCL 3 function)	■		693
409	Ridge center preset (FCL 3 function)	■		697
410	Preset inside rectangle	■		651
411	Preset outside rectangle	■		655
412	Preset from inside of circle (hole)	■		659
413	Preset from outside of circle (stud)	■		664
414	Preset from outside of corner	■		669
415	Preset from inside of corner	■		674
416	Preset circle center	■		679
417	Preset in TS axis	■		683
418	Preset from 4 holes	■		685
419	Preset in one selectable axis	■		690
420	Workpiece—measure angle	■		714
421	Measure circle outside (hole)	■		717
422	Measure circle inside (stud)	■		722
423	Workpiece—measure rectangle from inside	■		727
424	Workpiece—measure rectangle from outside	■		731
425	Measure inside width (slot)	■		734
426	Measure outside width (ridge)	■		737
427	Workpiece—measure in any selectable axis	■		740
430	Workpiece—measure bolt hole circle	■		743

Cycle number	Cycle name	DEF active	CALL active	Page
431	Workpiece—measure plane	■		746
441	Fast probing	■		764
450	KinematicsOpt: Save Kinematics (option)	■		806
451	KinematicsOpt: Measure Kinematics (option)	■		809
452	KinematicsOpt: Preset compensation	■		802
453	Kinematics grid	■		834
460	Calibrating the touch probe	■		776
461	Calibrate touch probe length	■		768
462	Calibrate touch probe inside radius	■		770
463	Calibrate touch probe outside radius	■		773
480	Calibrate TT	■		848
481	Measure/Inspect the tool length	■		850
482	Measure/Inspect the tool radius	■		854
483	Measure/Inspect the tool length and the tool radius	■		858
484	Calibrate TT	■		862
600	Global working space	■		790
601	Local working space	■		795
1410	Probing on edge	■		613
1411	Probing two circles	■		617
1420	Probing in plane	■		609

Index

2

2-D CODE... 251

3

3D Touch Probes... 590

A

About this manual... 44
Activate wheel edge... 578
Ascertain the load... 411
Automatic presetting
  Bolt circle... 679
  Center of 4 holes... 685
  Center of slot... 693
  Circular pocket (hole)... 659
  Circular stud... 664
  Fundamentals... 648
  Preset inside corner... 674
  Preset outside corner... 669
  rectangular pocket... 651
  rectangular stud... 655
  Ridge center... 697
  Single axis... 690
  Touch probe axis... 683

B

Basic rotation... 623
  Rot in rotary axis... 635
  Rot of 2 holes... 626
  Rot of 2 studs... 630
  Set directly... 644
Blank form update... 430

C

Calibration cycles... 766
  Calibration of TS... 776
  Calibration of TS in a ring... 770
  Calibration on stud... 773
  TS Calibration of tool length. 768
Camera-based check
  Basics... 782
Camera-based monitoring
  Global working space... 790
  Local working space... 795
Check unbalance... 450
Circular hole pattern... 246
Classification of results... 709
Contour cycles... 258
Coordinate transformation
  Axis-specific scaling factor... 231
  Datum shift... 219, 220
  Fundamentals... 218
  Mirroring... 226
  Presetting... 240
  Rotation... 228
  Scaling... 230
  Tilt working plane... 233
Coupling interpolation turning.. 364
Cycle... 62
  Calling... 64
  Define... 63
Cycles and point tables... 81
Cylinder surface
  Machine contour... 333
Cylinder surface cycles
  Contour... 333
  Cylinder surface... 323
  Fundamentals... 322
  Ridge... 330
  Slot... 326

D

DataMatrix code pattern... 251
Datum shift
  in program... 219
  with datum tables... 220
Determine inclined workpiece position
  Probing in plane... 609
  Probing on edge... 613
  Probing two circles... 617
  Set basic rotation... 644
  Touch probe cycles 14xx fundamentals... 601
  Touch probe cycles 4xx fundamentals... 622
Determining inclined workpiece position
  Basic rotation... 623
  Basic rotation over two holes... 626
  Basic rotation over two studs... 630
  Compensate basic rotation over a rotary axis... 635
  Rotation of C axis... 640
Dressing
  Diameter... 570
  Profile... 574
Drilling Cycles... 84
  Back boring... 98
  Bore milling... 110
  Boring... 89
  Centering... 121
  Drilling... 85
  Reaming... 87
  Single-lip deep-hole drilling.. 113
  Universal drilling... 92
  Universal pecking... 102
Dwell time... 357

E

Engraving... 382

F

Face milling... 388
Fast probing... 764
Feature content level... 51
Floor finishing... 275
FUNCTION TURNDATA... 430

G

Gear
  Definition... 395
  Fundamentals... 393
  Gear hobbing... 398
  Gear skiving... 404
Gear hobbing... 442
GLOBAL DEF... 68
Grinding
  General... 562
Grinding wheel
  Length compensation... 580
  Radius compensation... 582

I

Inclined workpiece position checking
  Fundamentals... 706
  Measure angle... 714
  Measure bolt hole circle... 743
  Measure circle... 722
  Measure coordinate... 740
  Measure hole... 717
  Measure plane... 746
  Measure rectangular pocket 727
  Measure rectangular stud... 731
  Measure ridge from outside 737
  Measure slot width... 734
  Polar preset... 713
  Reference plane... 712
Interpolation turning, contour finishing... 371

K

Kinematic measurement
  Accuracy... 814
  Backlash... 816
  Prerequisites... 804
Kinematics measurement
  Fundamentals... 802
  Hirth coupling... 812
  Kinematics grid... 834
  Measure kinematics... 809
  Preset compensation... 824
  Save kinematics... 806
KinematicsOpt... 802

M

Machining cycles... 166
  Circular pocket... 173
  Circular slot... 185

Circular stud ... 196
Face milling ... 206
Polygon stud ... 200
Rectangular pocket ... 167
Rectangular stud ... 191
Slot milling ... 180
Machining patterns ... 72
Measure circle from inside ... 717
Measure circle from outside ... 722
Measure inside width ... 734
Measure machine status ... 409
Measure ridge width ... 737
Measure slot width ... 734
Measuring
Angle ... 714
Bolt hole circle ... 743
Circle from outside ... 722
Coordinate ... 740
Holes ... 717
Inside width ... 734
Plane ... 746
Rectangle from inside ... 727
Rectangle from outside ... 731
Ridge width ... 737
using Cycle 3 ... 755
Measuring in 3-D ... 757

O

OCM
Contour data ... 307
Finishing floor ... 312
Finishing side ... 314
Roughing ... 309
OCM cycles ... 304

P

Parallel axis ... 67
PATTERN DEF
entering ... 73
using ... 73
Pattern definition with PATTERN DEF ... 72
frames ... 76
full circle ... 77
patterns ... 75
pitch circle ... 78
Point ... 74
Pecking ... 102
Point pattern
DataMatrix code ... 251
linear ... 249
polar ... 246
Point patterns ... 244
Point tables ... 79
Positioning logic ... 595
Probing feed rate ... 594
Probing in 3-D ... 759
Profile dressing ... 574
Program call
Via cycle ... 358

R

Reciprocating stroke
Define ... 565
Starting ... 568
Stopping ... 569
Recording measurement results ... 707
Rectangular pocket
Measuring ... 727
Rectangular stud
Measuring ... 731
Reference images ... 783

S

Side finishing ... 277
SL Cycles ... 258
3-D contour train ... 291
Contour ... 261
Contour data ... 266
Contour train ... 281
Contour train data ... 280
Floor finishing ... 275
Fundamentals ... 258
OCM contour data ... 307
OCM Finishing floor ... 312
OCM Finishing side ... 314
OCM fundamentals ... 304
OCM roughing ... 309
Pilot drilling ... 268
Roughing ... 270
Side finishing ... 277
Superimposed contours ... 262, 345
Trochoidal slot ... 285
with complex contour formula ... 340
with simple contour formula 351
Spindle orientation ... 359

T

Table of cycles ... 866
Fixed cycles ... 866
Grinding cycles ... 869
Touch probe cycles ... 870
Turning cycles ... 868
Tapping ... 128
Rigid tapping ... 132
with a floating tap holder ... 129
with chip breaking ... 136
Thread cutting ... 413
Thread milling
Fundamentals ... 141
Helical thread drilling/milling 155
internal threads ... 143
outside ... 159
thread drilling/milling ... 151
Thread milling/countersinking ... 147
Tilt working plane
Procedure ... 239
Tolerance ... 360
Tolerance monitoring ... 709
Tool compensation ... 710
Tool measurement
Calibrate TT ... 848
Calibrate TT 449 ... 862
Fundamentals ... 842
Machine parameters ... 844
Measure tool length and radius ... 858
Tool length ... 850
Tool radius ... 854
Tool monitoring ... 710
Tool table ... 846
Touch probe cycles 14xx
Evaluation of tolerances ... 607
Fundamentals ... 601
Probing in plane ... 609
Probing on edge ... 613
Probing two circles ... 617
Semi-automatic mode ... 603
Transfer an actual position ... 608
Touch probe data ... 597
Touch-probe table ... 596
Turning cycles ... 426, 453
Adjust XZ system ... 432
contour, longitudinal ... 466
Contour-parallel ... 470
Contour recessing, axial ... 532
Contour recessing, radial ... 528
Contour recess turning, axial ... 509
Contour recess turning, radial ... 505
Contour transverse ... 487
Expanded recess turning,axial ... 501
Extended recessing, axial ... 523
Extended recessing, radial ... 516
Plunge contour longitudinal ... 460
Plunge longitudinal extended ... 463
Plunge traverse, extended ... 483
Recessing, radial ... 513
Recess turning, expanded, radial ... 494
Recess turning, simple, radial dir ... 491
Reset rotary coordinate system ... 440
Shoulder, face ... 473
Shoulder, face extended ... 476

shoulder, longitudinal..... 454
shoulder, longitudinal extended..... 457
Simple recessing, axial..... 520
Simple recess turning, axial. 498
Simultaneous finishing..... 548
Thread, contour-parallel..... 544
Thread, extended..... 540
Thread, longitudinal..... 536
Traverse plunge..... 480

W

Working plane..... 233

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

www.klartext-portal.com
The Information Site for HEIDENHAIN Controls

Klartext App
The Klartext on Your Mobile Device

Google Play Store

Apple App Store

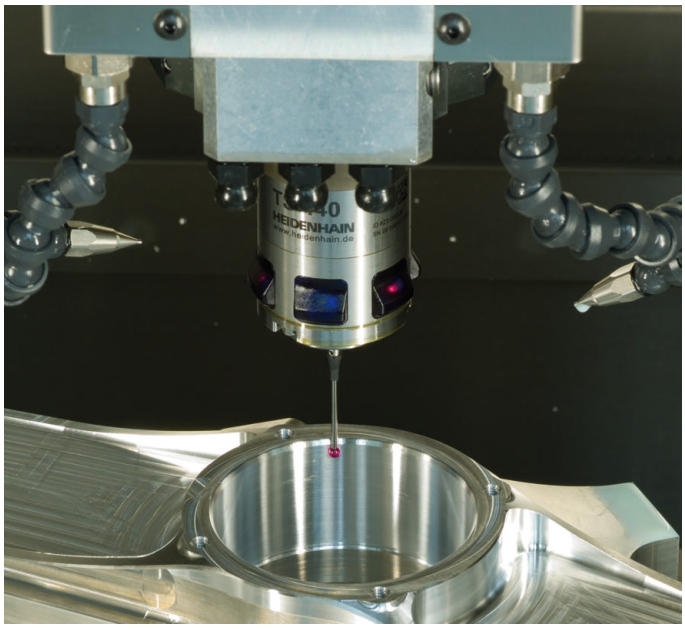
Touch probes from HEIDENHAIN

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	Infrared transmission
TS 642, TS 740	Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement



Tool touch probes

TT 160	Signal transmission by cable
TT 460	Infrared transmission

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
892905-28 · Ver08 · SW10 · 10/2019 · H · Printed in Germany