



HEIDENHAIN



TNC 620

User's Manual
Programming of Machining
Cycles

NC Software
817600-08
817601-08
817605-08

English (en)
01/2021

Contents

1 Fundamentals.....27

2 Fundamentals / Overviews.....41

3 Using Fixed Cycles.....45

4 Cycles: Drilling.....69

5 Cycles: Tapping / thread milling.....111

6 Cycles: Pocket Milling / Stud Milling / Slot Milling.....149

7 Cycles: Coordinate Transformations.....201

8 Cycles: Pattern definitions.....227

9 Cycles: Contour pocket.....241

10 Cycles: Optimized contour milling.....285

11 Cycles: Cylinder surface.....337

12 Cycles: Contour pocket with contour formula.....357

13 Cycles: Special Functions.....373

14 Tables of Cycles.....401

1	Fundamentals.....	27
1.1	About this manual.....	28
1.2	Control model, software, and features.....	30
	Software options.....	31
	New or modified cycle functions of software 81760x-08.....	36

2	Fundamentals / Overviews.....	41
2.1	Introduction.....	42
2.2	Available cycle groups.....	43
	Overview of fixed cycles.....	43
	Overview of touch probe cycles.....	44

3	Using Fixed Cycles.....	45
3.1	Working with fixed cycles.....	46
	Machine-specific cycles (option 19).....	46
	Defining a cycle using soft keys.....	47
	Defining a cycle using the GOTO function.....	47
	Calling a cycle.....	48
3.2	Program defaults for cycles.....	52
	Overview.....	52
	Entering GLOBAL DEF.....	52
	Using GLOBAL DEF information.....	53
	Global data valid everywhere.....	54
	Global data for drilling operations.....	54
	Global data for milling operations with pocket cycles.....	55
	Global data for milling operations with contour cycles.....	55
	Global data for positioning behavior.....	56
	Global data for probing functions.....	56
3.3	Pattern definition with PATTERN DEF.....	57
	Application.....	57
	Entering PATTERN DEF.....	58
	Using PATTERN DEF.....	58
	Defining individual machining positions.....	59
	Defining a single row.....	59
	Defining an individual pattern.....	60
	Defining an individual frame.....	61
	Defining a full circle.....	62
	Defining a pitch circle.....	63
3.4	Point tables.....	64
	Application.....	64
	Entering values into a point table.....	64
	Hiding single points from the machining process.....	65
	Selecting a point table in the NC program.....	65
	Calling a cycle in connection with point tables.....	66

4	Cycles: Drilling.....	69
4.1	Fundamentals.....	70
	Overview.....	70
4.2	DRILLING (Cycle 200, DIN/ISO: G200).....	72
	Application.....	72
	Please note while programming:.....	72
	Cycle parameters.....	73
4.3	REAMING (Cycle 201,DIN/ISO: G201, Option 19).....	74
	Application.....	74
	Please note while programming:.....	74
	Cycle parameters.....	75
4.4	BORING (Cycle 202, DIN/ISO: G202, option 19).....	76
	Application.....	76
	Please note while programming:.....	77
	Cycle parameters.....	79
4.5	UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203, option 19).....	80
	Application.....	80
	Please note while programming:.....	82
	Cycle parameters.....	83
4.6	BACK BORING (Cycle 204, DIN/ISO: G204, option 19).....	85
	Application.....	85
	Please note while programming:.....	86
	Cycle parameters.....	87
4.7	UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205, option 19).....	89
	Application.....	89
	Please note while programming:.....	90
	Cycle parameters.....	91
	Chip removal and chip breaking.....	93
4.8	BORE MILLING (Cycle 208, DIN/ISO: G208, option 19).....	95
	Application.....	95
	Please note while programming:.....	96
	Cycle parameters.....	97
4.9	SINGLE-LIP DEEP HOLE DRILLING (Cycle 241, DIN/ISO: G241, option 19).....	98
	Application.....	98
	Please note while programming:.....	99
	Cycle parameters.....	100
	Position behavior when working with Q379.....	102

4.10	CENTERING (Cycle 240, DIN/ISO: G240, option 19)	106
	Application	106
	Please note while programming:	106
	Cycle parameters	107
4.11	Programming Examples	108
	Example: Drilling cycles	108
	Example: Using drilling cycles in connection with PATTERN DEF	109

5	Cycles: Tapping / thread milling.....	111
5.1	Fundamentals.....	112
	Overview.....	112
5.2	TAPPING with a floating tap holder (Cycle 206, ISO: G206).....	113
	Application.....	113
	Please note while programming:.....	114
	Cycle parameters.....	115
5.3	TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207).....	116
	Application.....	116
	Please note while programming!.....	117
	Cycle parameters.....	118
	Retracting after a program interruption.....	119
5.4	TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209, option 19).....	120
	Application.....	120
	Please note while programming:.....	121
	Cycle parameters.....	122
	Retracting after a program interruption.....	123
5.5	Fundamentals of Thread Milling.....	124
	Prerequisites.....	124
5.6	THREAD MILLING (Cycle 262, DIN/ISO: G262, option 19).....	126
	Application.....	126
	Please note while programming:.....	127
	Cycle parameters.....	128
5.7	THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263, option 19).....	130
	Application.....	130
	Please note while programming:.....	131
	Cycle parameters.....	132
5.8	THREAD MILLING (Cycle 264, DIN/ISO: G264, option 19).....	134
	Application.....	134
	Please note while programming:.....	135
	Cycle parameters.....	136
5.9	HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265, option 19).....	138
	Application.....	138
	Please note while programming:.....	139
	Cycle parameters.....	140
5.10	OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267, option 19).....	142
	Application.....	142

Please note while programming:..... 143

Cycle parameters..... 144

5.11 Programming Examples..... 146

Example: Thread milling..... 146

6	Cycles: Pocket Milling / Stud Milling / Slot Milling.....	149
6.1	Fundamentals.....	150
	Overview.....	150
6.2	RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251, Option 19).....	151
	Application.....	151
	Please note while programming!.....	152
	Cycle parameters.....	154
	Plunging strategy Q366 with RCUTS.....	157
6.3	CIRCULAR POCKET (Cycle 252, DIN/ISO: G252, Option 19).....	158
	Application.....	158
	Please note while programming:.....	160
	Cycle parameters.....	162
	Plunging strategy Q366 with RCUTS.....	164
6.4	SLOT MILLING (Cycle 253, DIN/ISO: G253, Option 19).....	165
	Application.....	165
	Please note while programming:.....	166
	Cycle parameters.....	167
6.5	CIRCULAR SLOT (Cycle 254, DIN/ISO: G254, Option 19).....	169
	Application.....	169
	Please note while programming:.....	170
	Cycle parameters.....	172
6.6	RECTANGULAR STUD (Cycle 256, DIN/ISO: G256, Option 19).....	175
	Application.....	175
	Please note while programming:.....	176
	Cycle parameters.....	177
6.7	CIRCULAR STUD (Cycle 257, DIN/ISO: G257, Option 19).....	180
	Application.....	180
	Please note while programming:.....	181
	Cycle parameters.....	182
6.8	POLYGON STUD (Cycle 258, DIN/ISO: G258, Option 19).....	184
	Application.....	184
	Please note while programming:.....	185
	Cycle parameters.....	187
6.9	FACE MILLING (Cycle 233, DIN/ISO: G233, Option 19).....	190
	Application.....	190
	Please note while programming:.....	194
	Cycle parameters.....	195

6.10 Programming Examples.....	198
Example: Milling pockets, studs and slots.....	198

7	Cycles: Coordinate Transformations.....	201
7.1	Fundamentals.....	202
	Overview.....	202
	Effectiveness of coordinate transformations.....	202
7.2	DATUM SHIFT (Cycle 7, DIN/ISO: G54).....	203
	Application.....	203
	Please note while programming.....	203
	Cycle parameters.....	203
7.3	DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53).....	204
	Application.....	204
	Please note while programming:.....	205
	Cycle parameters.....	205
	Selecting a datum table in the part program.....	206
	Editing the datum table in the Programming mode of operation.....	206
	Editing the datum table in the Program Run, Single Block or Program Run, Full Sequence operating mode.....	208
	Configuring a datum table.....	208
	Leaving a datum table.....	209
	Status displays.....	209
7.4	MIRRORING (Cycle 8, DIN/ISO: G28).....	210
	Application.....	210
	Please note while programming:.....	210
	Cycle parameters.....	210
7.5	ROTATION (Cycle 10, DIN/ISO: G73).....	211
	Application.....	211
	Please note while programming:.....	212
	Cycle parameters.....	212
7.6	SCALING (Cycle 11, DIN/ISO: G72).....	213
	Application.....	213
	Cycle parameters.....	213
7.7	AXIS-SPECIFIC SCALING (Cycle 26).....	214
	Application.....	214
	Please note while programming:.....	214
	Cycle parameters.....	215
7.8	WORKING PLANE (Cycle 19, DIN/ISO: G80, Option 8).....	216
	Application.....	216
	Please note while programming:.....	217
	Cycle parameters.....	218
	Reset.....	219

Positioning the axes of rotation.....	219
Position display in a tilted system.....	220
Monitoring of the working space.....	220
Positioning in a tilted coordinate system.....	221
Combining coordinate transformation cycles.....	221
Procedure for working with Cycle 19 WORKING PLANE.....	222
7.9 PRESETTING (Cycle 247, DIN/ISO: G247).....	223
Application.....	223
Please note before programming:.....	223
Cycle parameters.....	223
Status displays.....	223
7.10 Programming Examples.....	224
Example: coordinate transformation cycles.....	224

8	Cycles: Pattern definitions.....	227
8.1	Fundamentals.....	228
	Overview.....	228
8.2	POLAR PATTERN (Cycle 220, DIN/ISO: G220, Option 19).....	230
	Application.....	230
	Please note while programming:.....	230
	Cycle parameters.....	231
8.3	CARTESIAN PATTERN (Cycle 221, DIN/ISO: G221, Option 19).....	233
	Application.....	233
	Please note while programming:.....	234
	Cycle parameters.....	235
8.4	DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224, Option 19).....	236
	Application.....	236
	Please note while programming!.....	237
	Cycle parameters.....	238
8.5	Programming Examples.....	239
	Example: Polar hole patterns.....	239

9	Cycles: Contour pocket.....	241
9.1	SL Cycles.....	242
	Fundamentals.....	242
	Overview.....	244
9.2	CONTOUR (Cycle 14, DIN/ISO: G37).....	245
	Application.....	245
	Cycle parameters.....	245
9.3	Superimposed contours.....	246
	Fundamentals.....	246
	Subprograms: overlapping pockets.....	246
	Area of inclusion.....	247
	Area of exclusion.....	248
	Area of intersection.....	249
9.4	CONTOUR DATA (Cycle 20, DIN/ISO: G120, option 19).....	250
	Application.....	250
	Cycle parameters.....	251
9.5	PILOT DRILLING (Cycle 21, DIN/ISO: G121, option 19).....	252
	Application.....	252
	Please note while programming:.....	253
	Cycle parameters.....	253
9.6	ROUGHING (Cycle 22, DIN/ISO: G122, option 19).....	254
	Application.....	254
	Please note while programming:.....	255
	Cycle parameters.....	256
9.7	FLOOR FINISHING (Cycle 23, DIN/ISO: G123, option 19).....	258
	Application.....	258
	Please note while programming:.....	259
	Cycle parameters.....	259
9.8	SIDE FINISHING (Cycle 24, DIN/ISO: G124, option 19).....	260
	Application.....	260
	Please note while programming:.....	261
	Cycle parameters.....	262
9.9	CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270, option 19).....	263
	Application.....	263
	Cycle parameters.....	264
9.10	CONTOUR TRAIN (Cycle 25, DIN/ISO: G125, option 19).....	265
	Application.....	265

Please note while programming!.....	266
Cycle parameters.....	267
9.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275, Option 19).....	269
Application.....	269
Please note while programming:.....	271
Cycle parameters.....	272
9.12 THREE-D CONT. TRAIN (Cycle 276, DIN/ISO: G276, option 19).....	274
Application.....	274
Please note while programming:.....	276
Cycle parameters.....	277
9.13 Programming Examples.....	279
Example: Roughing-out and fine-roughing a pocket.....	279
Example: Pilot drilling, roughing-out and finishing overlapping contours.....	281
Example: Contour train.....	283

10 Cycles: Optimized contour milling.....	285
10.1 OCM cycles (option 167).....	286
OCM fundamentals.....	286
Overview.....	289
10.2 OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167).....	290
Application.....	290
Please note while programming!.....	290
Cycle parameters.....	290
10.3 OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167).....	292
Application.....	292
Please note while programming!.....	293
Cycle parameters.....	294
10.4 OCM cutting data calculator (Option 167).....	296
Fundamentals of the OCM cutting data calculator.....	296
Operation.....	297
Fillable form.....	298
Process parameters.....	301
Achieving an optimum result.....	302
10.5 OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167).....	304
Application.....	304
Please note while programming!.....	304
Cycle parameters.....	305
10.6 OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167).....	307
Application.....	307
Please note while programming!.....	307
Cycle parameters.....	308
10.7 OCM CHAMFERING (Cycle 277, DIN/ISO: G277, Option 167).....	309
Application.....	309
Please note while programming!.....	310
Cycle parameters.....	311
10.8 OCM standard shapes.....	312
Fundamentals.....	312
10.9 OCM RECTANGLE (Cycle 1271, DIN/ISO: G1271, Option 167).....	313
Application.....	313
Please note while programming!.....	313
Cycle parameters.....	314

10.10 OCM CIRCLE (Cycle 1272, DIN/ISO: G1272, Option 167).....	316
Application.....	316
Please note while programming!.....	316
Cycle parameters.....	317
10.11 OCM SLOT / RIDGE (Cycle 1273, DIN/ISO: G1273, Option 167).....	318
Application.....	318
Please note while programming!.....	318
Cycle parameters.....	319
10.12 OCM POLYGON (Cycle 1278, DIN/ISO: G1278, Option 167).....	320
Application.....	320
Please note while programming!.....	320
Cycle parameters.....	321
10.13 OCM RECTANGLE BOUNDARY (Cycle 1281, DIN/ISO: G1281, Option 167).....	323
Application.....	323
Please note while programming!.....	323
Cycle parameters.....	324
10.14 OCM CIRCLE BOUNDARY (Cycle 1282, DIN/ISO: G1282, Option 167).....	325
Application.....	325
Please note while programming!.....	325
Cycle parameters.....	326
10.15 Programming examples.....	327
Example: Open pocket and fine roughing with OCM cycles.....	327
Example: Program various depths with OCM cycles.....	330
Example: Face milling and fine roughing with OCM cycles.....	332
Example: Contour with OCM shape cycles.....	334

11 Cycles: Cylinder surface.....	337
11.1 Fundamentals.....	338
Overview of cylindrical surface cycles.....	338
11.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, option 8).....	339
Application.....	339
Please note while programming:.....	340
Cycle parameters.....	341
11.3 CYLINDER SURFACE slot milling (Cycle 28, DIN/ISO: G128, Option 8).....	342
Application.....	342
Please note while programming:.....	344
Cycle parameters.....	345
11.4 CYLINDRICAL SURFACE ridge milling (Cycle 29, DIN/ISO: G129, Option 8).....	346
Application.....	346
Please note while programming:.....	347
Cycle parameters.....	348
11.5 CYLINDRICAL SURFACE CONTOUR (Cycle 39, DIN/ISO: G139, option 8).....	349
Application.....	349
Please note while programming:.....	351
Cycle parameters.....	352
11.6 Programming Examples.....	353
Example: Cylinder surface with Cycle 27.....	353
Example: Cylinder surface with Cycle 28.....	355

12 Cycles: Contour pocket with contour formula.....	357
12.1 SL or OCM cycles with complex contour formula.....	358
Fundamentals.....	358
Selecting an NC program with contour definitions.....	360
Defining contour descriptions.....	361
Entering a complex contour formula.....	362
Superimposed contours.....	363
Machining contours with SL or OCM cycles.....	365
Example: Roughing and finishing superimposed contours with the contour formula.....	366
12.2 SL or OCM cycles with simple contour formula.....	369
Fundamentals.....	369
Entering a simple contour formula.....	371
Contour machining with SL Cycles.....	372

13 Cycles: Special Functions.....	373
13.1 Fundamentals.....	374
Overview.....	374
13.2 DWELL TIME (Cycle 9, ISO: G04).....	375
Application.....	375
Cycle parameters.....	375
13.3 PROGRAM CALL (Cycle 12, ISO: G39).....	376
Application.....	376
Please note while programming:.....	376
Cycle parameters.....	376
13.4 SPINDLE ORIENTATION (Cycle 13, ISO: G36).....	377
Application.....	377
Please note while programming:.....	377
Cycle parameters.....	377
13.5 TOLERANCE (Cycle 32, DIN/ISO: G62).....	378
Application.....	378
Influences of the geometry definition in the CAM system.....	378
Please note while programming!.....	379
Cycle parameters.....	380
13.6 ENGRAVING (Cycle 225, ISO: G225).....	381
Application.....	381
Please note while programming:.....	381
Cycle parameters.....	382
Allowed engraving characters.....	384
Characters that cannot be printed.....	384
Engraving system variables.....	385
Engraving the name and path of an NC program.....	386
Engraving the counter reading.....	386
13.7 FACE MILLING (Cycle 232, DIN/ISO: G232, Option 19).....	387
Application.....	387
Please note while programming:.....	389
Cycle parameters.....	390
13.8 MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155).....	393
Application.....	393
Please note while programming!.....	394
Cycle parameters.....	394
13.9 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143).....	395
Application.....	395

Please note while programming:.....	396
Cycle parameters.....	397
13.10 THREAD CUTTING (Cycle 18, DIN/ISO: G86).....	398
Application.....	398
Please note while programming:.....	399
Cycle parameters.....	400

14	Tables of Cycles.....	401
14.1	Table of cycles.....	402
	Machining cycles.....	402

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 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 continuously strive 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 620	817600-08
TNC 620 E	817601-08
TNC 620 Programming station	817605-08

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 cycle functions not connected to the machining cycles in the **Programming of Measuring Cycles for Workpieces and Tools** User's Manual. This manual is available from HEIDENHAIN upon request.

ID of User's Manual for Programming of Measuring Cycles for Workpieces and Tools: 1303431-xx



User's Manual:

All control functions not related to the cycles are described in the TNC 620 User's Manual. This manual is available from HEIDENHAIN upon request.

Conversational Programming User's Manual ID: 1096883-xx

ISO Programming User's Manual ID: 1096887-xx

User's Manual for Setup, Testing and Running NC programs ID: 1263172-xx

Software options

The TNC 620 features various software options, each of which can be enabled separately by your machine tool builder. The respective options provide the functions listed below:

Additional Axis (option 0 and option 1)

Additional axis	Additional control loops 1 and 2
------------------------	----------------------------------

Advanced Function Set 1 (option 8)

Advanced functions (set 1)	Machining with rotary tables <ul style="list-style-type: none"> ■ 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)

Advanced functions (set 2) Export license required	3-D machining: <ul style="list-style-type: none"> ■ 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)
--	---

Touch Probe Functions (option 17)

Touch probe functions	Touch probe cycles: <ul style="list-style-type: none"> ■ Compensation of tool misalignment in automatic mode ■ Set the preset in the Manual operation mode of operation ■ Presetting in automatic mode ■ Automatically measuring workpieces ■ Tools can be measured automatically
------------------------------	--

HEIDENHAIN DNC (option 18)

Communication with external PC applications over COM component

Advanced Programming Features (option 19)

Expanded programming functions	FK free contour programming: Programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
---------------------------------------	---

Advanced Programming Features (option 19)

Fixed cycles:

- Peck drilling, reaming, boring, counterboring, centering
 - Milling internal and external threads
 - Milling of rectangular and circular pockets and studs
 - Clearing level and oblique surfaces
 - Milling of straight and circular slots
 - Circular and linear point patterns
 - Contour train, contour pocket, trochoidal contour slot
 - Engraving
 - OEM cycles (special cycles developed by the machine tool builder) can be integrated
-

Advanced Graphic Features (option 20)

Expanded graphic functions**Program-verification graphics, program-run graphics**

- Plan view
 - Projection in three planes
 - 3-D view
-

Advanced Function Set 3 (option 21)

Expanded functions Group 3**Tool compensation:**

M120: Radius-compensated contour look ahead for up to 99 NC blocks (LOOK AHEAD)

3-D machining:

M118: Superimpose handwheel positioning during program run

Pallet Management (option 22)

Pallet management

Processing workpieces in any sequence

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
-

KinematicsOpt (option 48)

Optimizing the machine kinematics

- Backup/restore active kinematics
 - Test active kinematics
 - Optimize active kinematics
-

OPC UA NC Server (1 to 6) (options 56 to 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

Extended Tool Management (option 93)

Extended tool management	Python-based
---------------------------------	--------------

Remote Desktop Manager (option 133)

Remote operation of external computer units	<ul style="list-style-type: none"> ■ Windows on a separate computer unit ■ Incorporated in the control's interface
--	--

State Reporting Interface – SRI (option 137)

HTTP accesses to the control status	<ul style="list-style-type: none"> ■ Reading out the times of status changes ■ Reading out the active NC programs
--	---

Cross Talk Compensation – CTC (option 141)

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

Position Adaptive Control – PAC (option 142)

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

Load Adaptive Control – LAC (option 143)

Adaptive load control	<ul style="list-style-type: none"> ■ Automatic determination of workpiece weight and frictional forces ■ Adaptation of the control parameters depending on the current mass of the workpiece
------------------------------	--

Active Chatter Control – ACC (option 145)

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

Machine Vibration Control – MVC (option 146)

Vibration damping for machines	<p>Damping of machine oscillations for improving the workpiece surface quality through the following functions:</p> <ul style="list-style-type: none"> ■ Active Vibration Damping (AVD) ■ Frequency Shaping Control (FSC)
---------------------------------------	---

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
--	---

Opt. contour milling (option 167)

Optimized contour cycles	Cycles for machining any pockets and islands using trochoidal milling
---------------------------------	---

Further options available



HEIDENHAIN offers further hardware enhancements and software options that can be configured and implemented only by your machine tool builder. This includes functional safety (FS), for example.

For more information, please refer to your machine tool builder's documentation or the HEIDENHAIN brochure titled **Options and Accessories**.

ID: 827222-xx

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

The control software contains open-source software that is subject to special terms of use. These special terms of use have priority.

Further information is available on the control as follows:

- ▶ Press the **MOD** key to open the **Settings and information** dialog
- ▶ Select **Code-number entry** in the dialog
- ▶ Press the **LICENSE INFO** soft key or select **Settings and information** → **General information** → **License info** directly in the dialog

Furthermore, the control software contains binary libraries of the OPC UA software from Softing Industrial Automation GmbH. For these libraries, additionally and preferentially the terms of use agreed between HEIDENHAIN and Softing Industrial Automation GmbH shall apply.

When using the OPC UA NC server or DNC server, you can influence the behavior of the control. Therefore, before using these interfaces for productive purposes, verify that the control can still be operated without malfunctions or drops in performance. The manufacturer of the software that uses these communication interfaces is responsible for performing system tests.

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, they are always provided at the end of the cycle definition. The section "New or modified cycle functions of software 81760x-08" gives you an overview of the optional Q parameters that have been added in this software version. You can decide for yourself whether you would like to define optional Q parameters or delete them with the NO ENT key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing 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

or

- ▶ Enter a 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 620. 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 backward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

New or modified cycle functions of software 81760x-08



Overview of new and modified software functions

Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.
ID: 1322094-xx

User's Manual for Programming of Machining Cycles:

New functions:

- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167)
This cycle allows the control to deburr contours that were last defined, roughed, or finish-machined with other OCM cycles.
Further information: "OCM CHAMFERING (Cycle 277, DIN/ISO: G277, Option 167) ", Page 309
- Cycle **1271 OCM RECTANGLE** (ISO: **G1271**, option 167)
This cycle allows you to define a rectangle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
Further information: "OCM RECTANGLE (Cycle 1271, DIN/ISO: G1271, Option 167) ", Page 313
- Cycle **1272 OCM CIRCLE** (ISO: **G1272**, option 167)
This cycle allows you to define a circle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
Further information: "OCM CIRCLE (Cycle 1272, DIN/ISO: G1272, Option 167) ", Page 316
- Cycle **1273 OCM SLOT / RIDGE** (ISO: **G1273**, option 167)
This cycle allows you to define a slot that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
Further information: "OCM SLOT / RIDGE (Cycle 1273, DIN/ISO: G1273, Option 167) ", Page 318
- Cycle **1278 OCM POLYGON** (ISO: **G1278**, option 167)
This cycle allows you to define a polygon that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
Further information: "OCM POLYGON (Cycle 1278, DIN/ISO: G1278, Option 167) ", Page 320

- Cycle **1281 OCM RECTANGLE BOUNDARY** (ISO: **G1281**, option 167)

This cycle allows you to define a rectangular boundary for islands or open pockets that you previously programmed with the standard OCM forms.

Further information: "OCM RECTANGLE BOUNDARY (Cycle 1281, DIN/ISO: G1281, Option 167) ", Page 323

- Cycle **1282 OCM CIRCLE BOUNDARY** (ISO: **G1282**, option 167)

This cycle allows you to define a circular boundary for islands or open pockets that you previously programmed with the standard OCM forms.

Further information: "OCM CIRCLE BOUNDARY (Cycle 1282, DIN/ISO: G1282, Option 167) ", Page 325

- The control offers an **OCM cutting data calculator** with which you can determine the optimum cutting data for the **272 OCM ROUGHING** cycle (ISO: **G272**, option 167). Press the **OCM CUTTING DATA** soft key to open the cutting data calculator during cycle definition. You can load the results directly into the cycle parameters.

Further information: "OCM cutting data calculator (Option 167)", Page 296

Modified functions:

- Cycle **225 ENGRAVING** (ISO: **G225**) allows you to use system variables to engrave the number of the current week.
Further information: "Engraving system variables", Page 385
- Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**, option 19) restore the spindle status after machining to that which was active before the cycle.
Further information: "BORING (Cycle 202, DIN/ISO: G202, option 19)", Page 76
Further information: "BACK BORING (Cycle 204, DIN/ISO: G204, option 19)", Page 85
- The threads of cycles **206 TAPPING** (ISO: **G206**), **207 RIGID TAPPING** (ISO: **G207**), **209 TAPPING W/ CHIP BRKG** (ISO: **G209**, option 19), and **18 THREAD CUTTING** (ISO: **G18**) are shown shaded in Test Run mode.
- If the defined usable length in column **LU** of the tool table is less than the depth, the control displays an error message.
The following cycles monitor the usable length **LU**:
 - All cycles for drilling and boring
 - All cycles for tapping
 - All cycles for the machining of pockets and studs
 - Cycle 22 **ROUGH-OUT** (ISO: **G122**, option 19)
 - Cycle 23 **FLOOR FINISHING** (ISO: **G123**, option 19)
 - Cycle 24 **SIDE FINISHING** (ISO: **G124**, option 19)
 - Cycle 233 **FACE MILLING** (ISO: **G233**, option 19)
 - Cycle 272 **OCM ROUGHING** (ISO: **G272**, option 167)
 - Cycle 273 **OCM FINISHING FLOOR** (ISO: **G273**, option 167)
 - Cycle 274 **OCM FINISHING SIDE** (ISO: **G274**, option 167)
- Cycles **251 RECTANGULAR POCKET** (ISO: **G251**), **252 CIRCULAR POCKET** (ISO: **G252**, option 19), and **272 OCM ROUGHING** (ISO: **G272**, option 167) take into consideration a cutting width defined in the **RCUTS** column when calculating the plunging path.
Further information: "RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251, Option 19)", Page 151
Further information: "CIRCULAR POCKET (Cycle 252, DIN/ISO: G252, Option 19)", Page 158
Further information: "OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167) ", Page 292
- Cycles **208 BORE MILLING** (ISO: **G208**), **253 SLOT MILLING** (ISO: **G208**), and **254 CIRCULAR SLOT** (ISO: **G254**, option 19) monitor a cutting width defined in the **RCUTS** column of the tool table. If the center of a tool that is not a center-cut tool would contact the workpiece surface, the control issues an error message.
Further information: "BORE MILLING (Cycle 208, DIN/ISO: G208, option 19)", Page 95
Further information: "SLOT MILLING (Cycle 253, DIN/ISO: G253, Option 19)", Page 165
Further information: "CIRCULAR SLOT (Cycle 254, DIN/ISO: G254, Option 19)", Page 169

- The machine manufacturer can hide Cycle **238 MEASURE MACHINE STATUS** (ISO: **G238**, option 155).
Further information: "MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)", Page 393
- An input value of 2 has been added to parameter **Q569 OPEN BOUNDARY** in Cycle **271 OCM CONTOUR DATA** (ISO: **G271**, option 167). The control uses this value to interpret the first contour within the **CONTOUR DEF** function as a boundary block of a pocket.
Further information: "OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167) ", Page 290
- Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167) was expanded:
 - The parameter **Q576 SPINDLE SPEED** allows you to define a spindle speed for the roughing tool.
 - The parameter **Q579 PLUNGING FACTOR S** allows you to define a factor for the spindle speed during plunging.
 - Use the parameter **Q575 INFEEED STRATEGY** to define whether the control machines the contour from top to bottom or vice versa.
 - The maximum input range of parameter **Q370 TOOL PATH OVERLAP** has been changed. Old: from 0.01 to 1. New: from 0.04 to 1.99.
 - If helical plunging is not possible, the control attempts a reciprocating plunging motion with the tool.**Further information:** "OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167) ", Page 292
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167) was expanded.
The following parameters have been added:
 - **Q595 STRATEGY:** machining with equal path distances or constant tool angle
 - **Q577 APPROACH RADIUS FACTOR:** factor for the tool radius in order to adapt the approach radius**Further information:** "OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167)", Page 304

User's Manual for Programming of Measuring Cycles for Workpieces and Tools:

Changed Functions

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**, option 17) can be used to calibrate a tool touch probe with cuboid probe contacts.
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**, option 17) first measures the tool length and then the tool radius.
- By default, Cycles **1410 PROBING ON EDGE** (ISO: **G1410**) and **1411 PROBING TWO CIRCLES** (ISO: **G1411**, option 17) calculate the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the cycles calculate the basic rotation in the workpiece coordinate system (W-CS).

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 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



- Press the **CYCL DEF** key

Soft key	Cycle group	Page
	Cycles for pecking, reaming, boring and counterboring	70
	Cycles for tapping, thread cutting and thread milling	112
	Cycles for milling pockets, studs, slots, and face milling	150
	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	202
	SL (Subcontour List) cycles for machining contours that consist of multiple overlapping subcontours as well as cycles for cylinder surface machining and trochoidal milling	244
	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	228
	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, determining the load,	374











- If required, switch to machine-specific machining cycles

The machine tool builder can integrate these types of machining cycles.

Overview of touch probe cycles



- Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic workpiece presetting	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic workpiece inspection	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Special cycles	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Touch probe calibration	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic kinematics measurement	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	Further information: User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	► 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 (option 19)



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

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

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



Programming note:

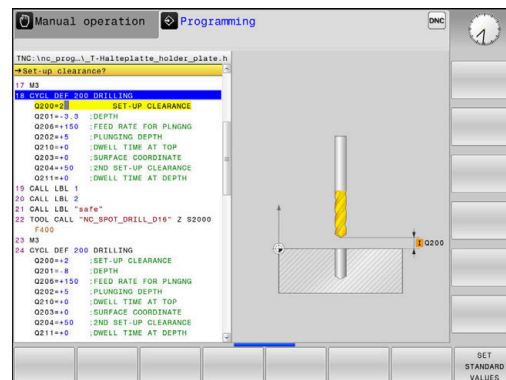
- Only program a DEF-active cycle between the definition of a CALL-active cycle and the cycle call if there are no interferences of transfer parameters of these two cycles.

Further information: "Calling a cycle", Page 48

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 desired cycle, e.g. Cycle **262 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 displays a pop-up window with an overview of the cycles.
- ▶ Select the desired cycle with the cursor keys
- or
- ▶ 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 (only required for test graphics)
- Tool call
- Spindle direction of rotation (**M3/M4** miscellaneous function)
- 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 **9 DWELL TIME**
- Cycle **12 PGM CALL**
- Cycle **13 ORIENTATION**
- Cycle **14 CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **32 TOLERANCE**
- Cycle **220 POLAR PATTERN**
- Cycle **221 CARTESIAN PATTERN**
- Cycle **224 DATAMATRIX CODE PATTERN**
- Cycle **238 MEASURE MACHINE STATUS**
- Cycle **239 ASCERTAIN THE LOAD**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **1271 OCM RECTANGLE**
- Cycle **1272 OCM CIRCLE**
- Cycle **1273 OCM SLOT / RIDGE**
- Cycle **1278 OCM POLYGON**
- Cycle **1281 OCM RECTANGLE BOUNDARY**
- Cycle **1282 OCM CIRCLE BOUNDARY**
- Cycles for coordinate transformation
- 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

**CYCLE
CALL
M**

- ▶ 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 57

Further information: "Point tables", Page 64

Calling a cycle with CYCL CALL POS

The **CYCL CALL POS** function calls the most recently defined canned 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



Programming and operating note:

- 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 machining 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.

or

- ▶ 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:

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

Calling an NC program as a cycle

- 
 - ▶ Press the **CYCL CALL** key
 - ▶ Press the soft key for the cycle call
- or
- ▶ Program **M99**



Programming and operating note:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
- 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 starting 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.

3.2 Program defaults for cycles

Overview

Some cycles 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 cycles used in the NC program. In the respective cycle you then simply reference the value defined at the beginning of the program.

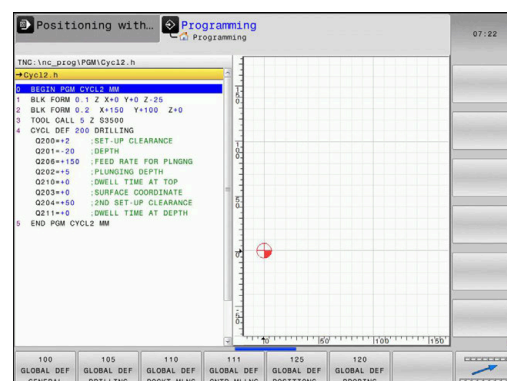
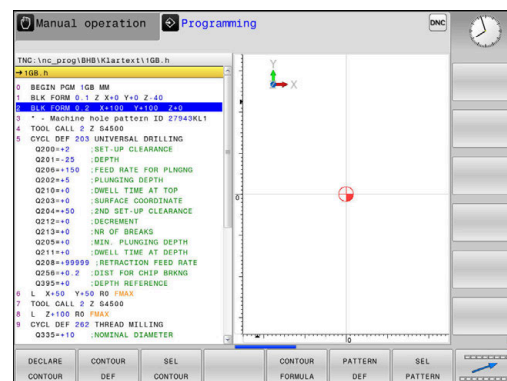
The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	54
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	54
110 GLOBAL DEF POCKET MILLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	55
111 GLOBAL DEF CNTR MILLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	55
125 GLOBAL DEF POSITIONING.	GLOBAL DEF POSITIONING Definition of the positioning behavior for CYCL CALL PAT	56
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	56

Entering GLOBAL DEF

Proceed as follows:






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

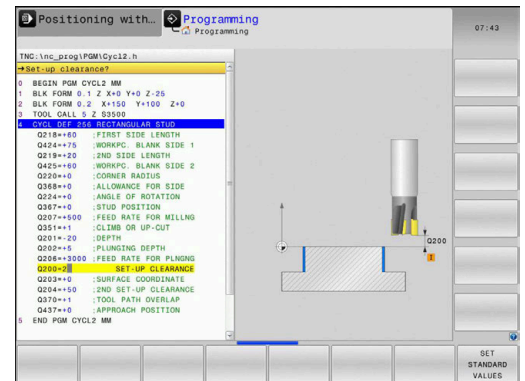


Using GLOBAL DEF information

If you entered the corresponding GLOBAL DEF functions at program start, you can reference these globally valid values for the definition of any cycle.

Proceed as follows:

- 
 - Press the **PROGRAMMING** key
- 
 - Press the **CYCL DEF** key
- 
 - Select the desired cycle group, e.g. pockets / studs / slot cycles
- 
 - Select the desired cycle, e.g. **RECTANGULAR STUD**
 - 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 you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

Global data valid everywhere

The parameters are valid for all machining cycles **2xx** and touch probe cycles **451, 452**

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value.
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
- ▶ **Q253 Feed rate for pre-positioning?:** Feed rate at which the control traverses the tool within a cycle.
Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- ▶ **Q208 Feed rate for retraction?:** Feed rate at which the control retracts the tool. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**

Example

11 GLOBAL DEF 100 GENERAL	
Q200=2	;SET-UP CLEARANCE
Q204=100	;2ND SET-UP CLEARANCE
Q253=+750	;F PRE-POSITIONING
Q208=+999	;RETRACTION FEED RATE

Global data for drilling operations

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

- ▶ **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking.
Input range: 0 to 99999.9999
- ▶ **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
- ▶ **Q211 Dwell time at the depth?:** Time in seconds that the tool remains at the hole bottom.
Input range: 0 to 3600.0000

Example

11 GLOBAL DEF 105 DRILLING	
Q256=+0.2	;DIST FOR CHIP BRKNG
Q210=+0	;DWELL TIME AT TOP
Q211=+0	;DWELL TIME AT DEPTH

Global data for milling operations with pocket cycles

The parameters apply to the cycles **208, 232, 233, 251 to 258, 262 to 264, 267, 272, 273, 275, and 277**

- ▶ **Q370 Path overlap factor?:** **Q370** x tool radius = stepover factor k. Input range: 0.1 to 1.9999
- ▶ **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)
- ▶ **Q366 Plunging strategy (0/1/2)?:** Type of plunging strategy:
 - 0:** vertical plunging. The control plunges the tool vertically, 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.

Example

11 GLOBAL DEF 110 POCKET MILLING	
Q370=+1	;TOOL PATH OVERLAP
Q351=+1	;CLIMB OR UP-CUT
Q366=+1	;PLUNGE

Global data for milling operations with contour cycles

The parameters apply to the cycles **20, 24, 25, 27 to 29, 39, and 276**

- ▶ **Q2 Path overlap factor?:** **Q2** x tool radius = stepover factor k. Input range: +0.0001 to 1.9999
- ▶ **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
- ▶ **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

Example

11 GLOBAL DEF 111 CONTOUR MILLING	
Q2=+1	;TOOL PATH OVERLAP
Q6=+2	;SET-UP CLEARANCE
Q7=+50	;CLEARANCE HEIGHT
Q9=+1	;ROTATIONAL DIRECTION

Global data for positioning behavior

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

- ▶ **Q345 Select positioning height (0/1):** Retraction in the tool axis at the end of a machining step, return to 2nd set-up clearance or to the position at the beginning of the unit

Example

```
11 GLOBAL DEF 125 POSITIONING
```

```
Q345=+1 ;SELECT POS. HEIGHT
```

Global data for probing functions

The parameters are valid for all touch probe cycles **4xx** and **14xx** as well as for Cycles **271, 1271, 1272, 1273, 1278**

- ▶ **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 at measuring height between measuring points
1: Move at clearance height between measuring points

Example

```
11 GLOBAL DEF 120 PROBING
```

```
Q320=+0 ;SET-UP CLEARANCE
```

```
Q260=+100 ;CLEARANCE HEIGHT
```

```
Q301=+1 ;MOVE TO CLEARANCE
```


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	59
	ROW Definition of a single row, straight or rotated	59
	PATTERN Definition of a single pattern, straight, rotated or distorted	60
	FRAME Definition of a single frame, straight, rotated or distorted	61
	CIRCLE Definition of a full circle	62
	PITCH CIRCLE Definition of a pitch circle	63

Entering PATTERN DEF

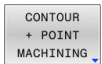
Proceed as follows:



- ▶ 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 48

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



Programming and operating note:

- A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
- 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 **Q345=1**. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.



Operating note:

- 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

Defining individual machining positions



Programming and operating notes:

- 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 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

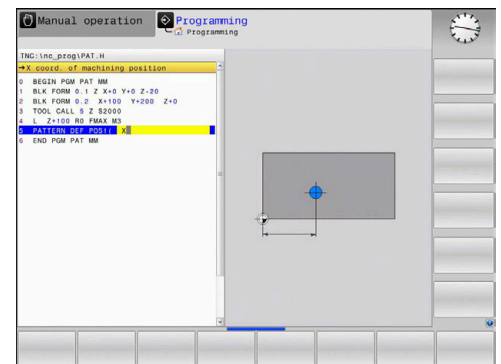
Example

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)

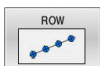


Defining a single row



Programming and operating notes:

- 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.



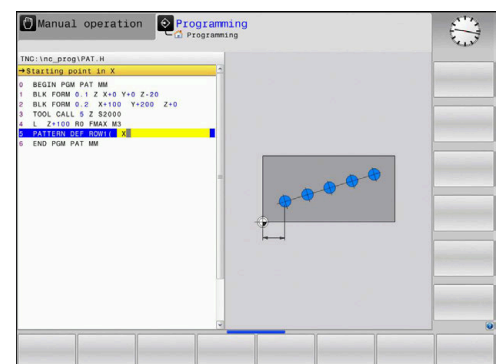
- ▶ **Starting point in X** (absolute): Coordinate of the pattern row starting point in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the pattern row starting point in the Y axis
- ▶ **Spacing of machining positions** (incremental): Distance between the machining positions. You can 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)

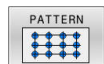


Defining an individual pattern



Programming and operating notes:

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

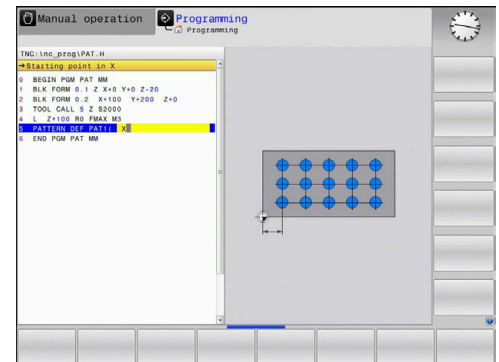


- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions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

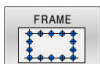


Defining an individual frame



Programming and operating notes:

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

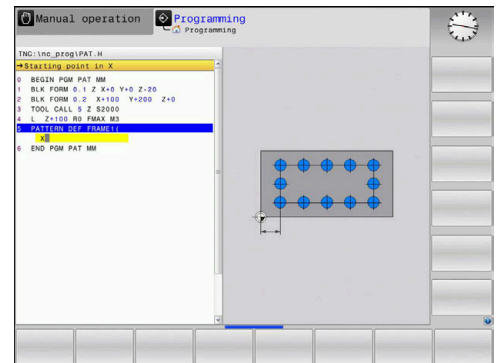


- ▶ **Starting point in X** (absolute): Coordinate of the frame starting point in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the frame starting point in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction.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



Programming and operating notes:

- 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.

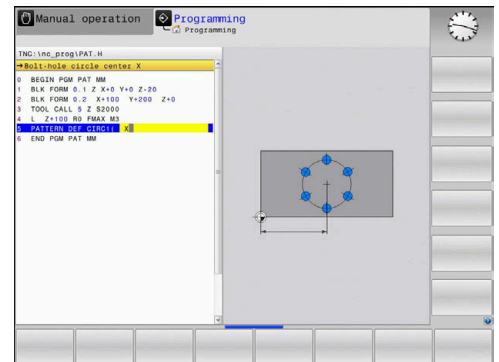


- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- ▶ **Bolt-hole circle diameter**: Diameter of the circular hole pattern
- ▶ **Starting angle**: Polar angle of the first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



Programming and operating notes:

- 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.

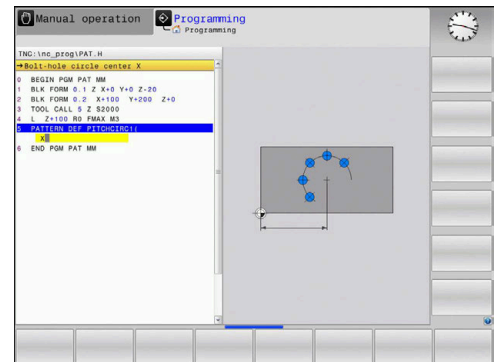


- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- ▶ **Bolt-hole circle diameter**: Diameter of the circular hole pattern
- ▶ **Starting angle**: Polar angle of the first 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.
- 
 - ▶ Press the **SORT/ HIDE COLUMNS** soft key, if necessary
 - > The control displays the desired coordinates or changes their order.



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

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

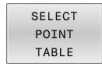
Selecting a point table in the NC program

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.



If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

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

or

- ▶ Press the **F MAX** soft key
- ▶ The control will use this feed rate to traverse between the points.
- ▶ 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 **Q345=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.



Programming and operating notes:

- 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




Cycles: Drilling

4.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	DRILLING (Cycle 200, DIN/ISO: G200) <ul style="list-style-type: none"> ■ Basic hole ■ Input of the dwell time at top and bottom ■ Depth reference selectable 	72
	REAMING (Cycle 201, DIN/ISO: G201, Option 19) <ul style="list-style-type: none"> ■ Reaming a hole ■ Input of the dwell time at bottom 	74
	BORING (Cycle 202, DIN/ISO: G202, option 19) <ul style="list-style-type: none"> ■ Boring a hole ■ Input of the retraction feed rate ■ Input of the dwell time at bottom ■ Input of the retracting movement 	76
	UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203, option 19) <ul style="list-style-type: none"> ■ Degression – hole with decreasing infeed ■ Input of the dwell time at top and bottom ■ Input of chip breaking behavior ■ Depth reference selectable 	80
	BACK BORING (Cycle 204, DIN/ISO: G204, option 19) <ul style="list-style-type: none"> ■ Machining a counterbore on the underside of the workpiece ■ Input of the dwell time ■ Input of the retracting movement 	85
	UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205, option 19) <ul style="list-style-type: none"> ■ Degression – hole with decreasing infeed ■ Input of chip breaking behavior ■ Input of a recessed starting point ■ Input of an advanced stop distance 	89

Soft key	Cycle	Page
	BORE MILLING (Cycle 208, DIN/ISO: G208, option 19) <ul style="list-style-type: none"> ■ Milling of a hole ■ Input of a pre-drill diameter ■ Climb or up-cut milling selectable 	95
	SINGLE-LIP DEEP HOLE DRILLING (Cycle 241, DIN/ISO: G241, option 19) <ul style="list-style-type: none"> ■ Drilling with single-lip deep hole drill ■ Recessed starting point ■ Direction of rotation and rotational speed for moving into and retracting from the hole ■ Input of the dwell depth 	98
	CENTERING (Cycle 240, DIN/ISO: G240, option 19) <ul style="list-style-type: none"> ■ Drilling a center hole ■ Input of the centering diameter or depth ■ Input of the dwell time at bottom 	106

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

Application

With this cycle, you can drill basic holes. In this cycle, the depth reference is selectable.

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 set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to 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 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

- 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

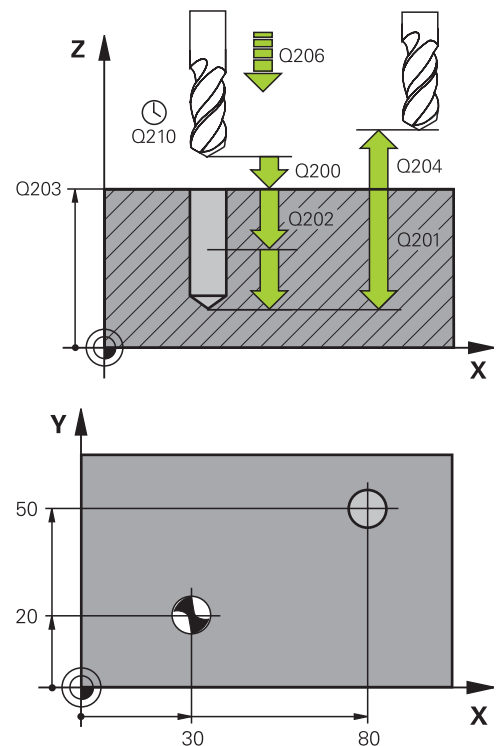


If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

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 99999.999
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 with reference to the active preset
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, Option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

With this cycle, you can machine basic fits. In this cycle, you can optionally define an optional dwell time at the bottom of the hole.

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 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!

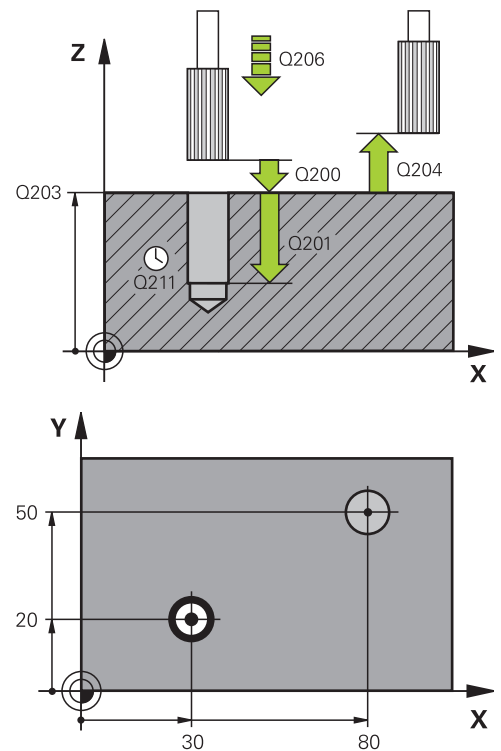
- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

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

With this cycle, you can bore holes. In this cycle, you can optionally define an optional dwell time at the bottom of the hole.

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 Then, the control moves the tool at the retraction feed rate to set-up clearance
- 7 The tool is again centered in the hole
- 8 The control restores the spindle status as it was at the cycle start.
- 9 If programmed, the control moves the tool at **FMAX** to 2nd set-up clearance. 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

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 programming an oriented spindle stop with reference to the angle entered 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.

NOTICE**Danger of collision!**

If you have activated **M136**, the tool will not move to the programmed set-up clearance once the machining operation is finished. The spindle rotation will stop at the bottom of the hole which, in turn, also stops the feed motion. There is a danger of collision as tool will not be retracted!

- ▶ Use **M137** to deactivate **M136** before the cycle start

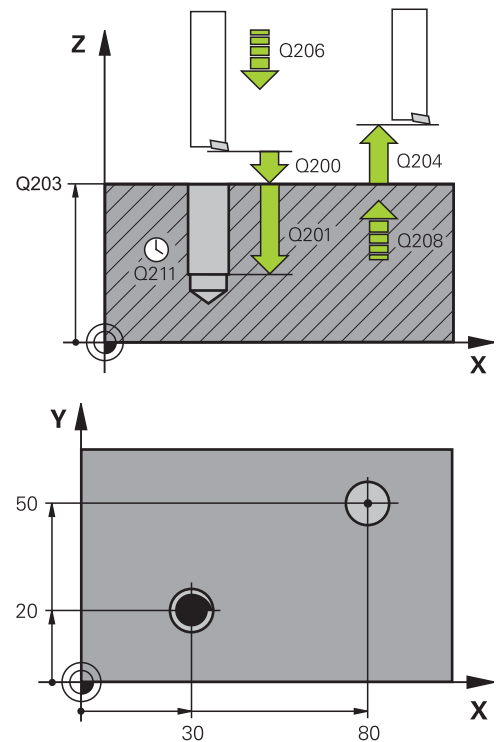
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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 with reference to the active preset
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 retract the tool
1: Disengage the tool in the minus direction of the principal axis
2: Disengage the tool in the minus direction of the principal axis
3: Disengage the tool in the minus direction of the principal axis
4: Disengage the tool in the minus direction of the principal 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, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

With this cycle, you can drill holes with decreasing infeed. In this cycle, you can optionally define an optional dwell time at the bottom of the hole. The cycle may be executed with or without chip breaking.

Cycle run

Behavior without chip breaking, 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 breaking, the control removes the tool from the hole after each infeed at the **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200**. It 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 **SET-UP CLEARANCE Q200** or to **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, without decrement:

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified **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 **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 **SET-UP CLEARANCE Q200** or to **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, with decrement

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

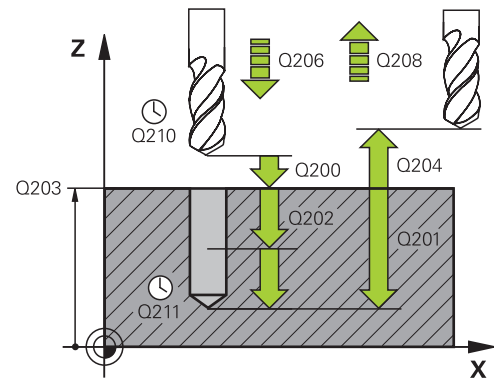
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ 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
- 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.
 - This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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 99999.999
 - 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 with reference to the active preset
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 to 99999.9999
- ▶ **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, option 19)

Application



Refer to your machine manual!

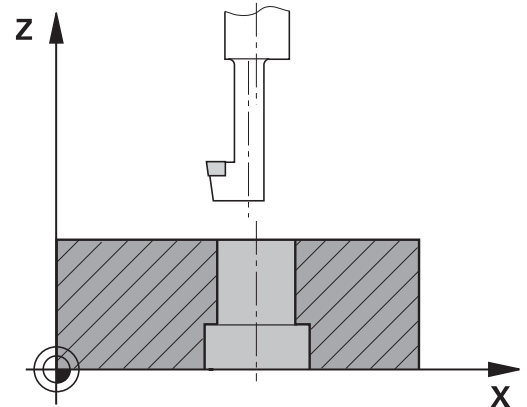
This function must be enabled and adapted by the machine tool builder.

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



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

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



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 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, if applicable, the coolant and moves the tool 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 moves at **FMAX** to set-up clearance.
- 7 The tool is again centered in the hole
- 8 The control restores the spindle status as it was at the cycle start.
- 9 If necessary, the control moves the tool to 2nd set-up 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!**

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 programming an oriented spindle stop with reference to the angle entered 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.

- 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: If you enter a positive sign, the tool bores in the direction of the positive spindle axis.
- 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.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF COUNTERBORE Q249**, the control will display an error message.

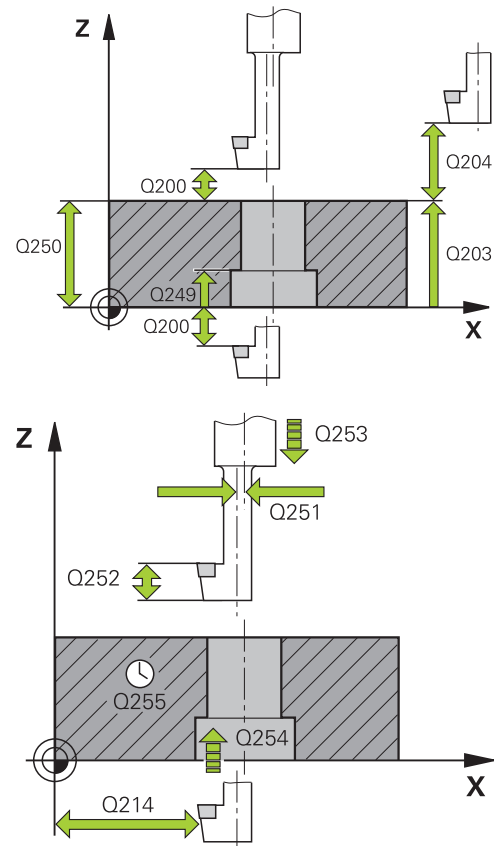


Enter the tool length measured up to the lower edge of the boring bar, not the cutting edge.

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 with reference to the active preset
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
Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE

- ▶ **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

Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

4.7 UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

With this cycle, you can drill holes with decreasing infeed. It is possible to enter a recessed starting point. In this cycle, you can optionally define a dwell time at the hole bottom. The cycle may be executed with or without chip breaking.

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 entered a recessed starting point, the control moves the tool at the defined positioning feed rate to 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 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!**

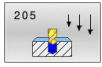
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ 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
- 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.
 - If **Q257 DEPTH FOR CHIP BRKNG** is greater than **Q202 PLUNGING DEPTH**, the operation is executed without chip breaking.
 - This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

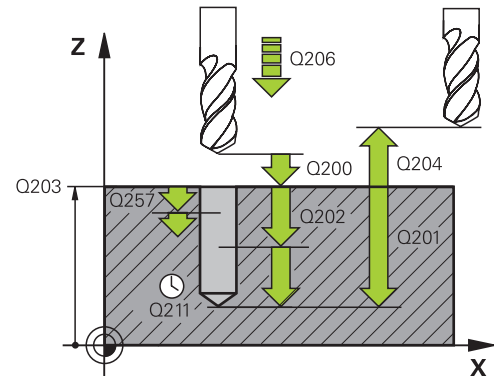


this cycle is not suitable for overlong drills. For overlong drills, use Cycle **241 SINGLE-LIP D.H.DRLNG**.

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 99999.999
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 with reference to the active preset
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 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
- ▶ **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

Chip removal and chip breaking

Chip removal

Chip removal depends on cycle parameter **Q202 PLUNGING DEPTH**.

When cycle parameter **Q202 PLUNGING DEPTH** is reached, the control starts chip removal. This means that the control always moves the tool to the retraction height, irrespective of the recessed starting point **Q379**. The retraction height results from **Q200 SET-UP CLEARANCE + Q203 SURFACE COORDINATE**

Example:

0 BEGIN PGM 205 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	Retracting the tool
5 CYCL DEF 205 UNIVERSAL PECKING	Cycle definition
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+250 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q212=+0 ;DECREMENT	
Q205=+0 ;MIN. PLUNGING DEPTH	
Q258=+0.2 ;UPPER ADV STOP DIST	
Q259=+0.2 ;LOWER ADV STOP DIST	
Q257=+0 ;DEPTH FOR CHIP BRKNG	
Q256=+0.2 ;DIST FOR CHIP BRKNG	
Q211=+0.2 ;DWELL TIME AT DEPTH	
Q379=+10 ;STARTING POINT	
Q253=+750 ;F PRE-POSITIONING	
Q208=+3000 ;RETRACTION FEED RATE	
Q395=+0 ;DEPTH REFERENCE	
6 L X+30 Y+30 R0 FMAX M3	Approach hole, spindle ON
7 CYCL CALL	Cycle call
11 L Z+250 R0 FMAX M30	Retract the tool, end program
12 END PGM 205 MM	

Chip breaking

Chip breaking depends on cycle parameter **Q257 DEPTH FOR CHIP BRKNG**.

When cycle parameter **Q257 DEPTH FOR CHIP BRKNG** is reached, the control starts chip breaking. This means that the control retracts the tool by the value defined in **Q256 DIST FOR CHIP BRKNG**. Chip removal starts once the tool reaches the **PLUNGING DEPTH**. The entire process is repeated until **Q202 DEPTH** is reached.

Example:

0 BEGIN PGM 205 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	Retracting the tool
5 CYCL DEF 205 UNIVERSAL PECKING	Cycle definition
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+250 ;FEED RATE FOR PLNGNG	
Q202=+10 ;PLUNGING DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q212=+0 ;DECREMENT	
Q205=+0 ;MIN. PLUNGING DEPTH	
Q258=+0.2 ;UPPER ADV STOP DIST	
Q259=+0.2 ;LOWER ADV STOP DIST	
Q257=+3 ;DEPTH FOR CHIP BRKNG	
Q256=+0.5 ;DIST FOR CHIP BRKNG	
Q211=+0.2 ;DWELL TIME AT DEPTH	
Q379=+0 ;STARTING POINT	
Q253=+750 ;F PRE-POSITIONING	
Q208=+3000 ;RETRACTION FEED RATE	
Q395=+0 ;DEPTH REFERENCE	
6 L X+30 Y+30 R0 FMAX M3	Approach hole, spindle ON
7 CYCL CALL	Cycle call
11 L Z+250 R0 FMAX M30	Retract the tool, end program
12 END PGM 205 MM	

4.8 BORE MILLING (Cycle 208, DIN/ISO: G208, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

With this cycle, you can mill holes. In this cycle, you can define an optional, pre-drilled diameter.

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 helical path from the current position to the entered drilling 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 centers the tool in the hole again and retracts it to 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 set-up clearance or to 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**



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.

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**Caution: Danger to the workpiece and tool!**

If the selected infeed is too large, there is a danger of tool breakage and damage to the workpiece.

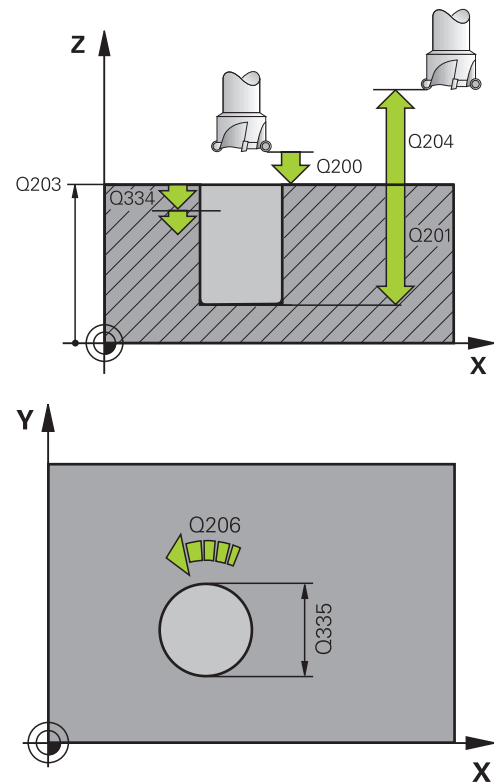
- ▶ Specify the maximum possible plunge angle and the corner radius **DR2** in the **ANGLE** column of the **TOOL.T** tool table.
- The control automatically calculates the max. permissible infeed and changes your entered value accordingly, if necessary.

- 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.
- When calculating the overlap factor, the control takes the corner radius **DR2** of the current tool into account.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

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 with reference to the active preset
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 pre-drilled 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, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Cycle **241 SINGLE-LIP D.H.DRLNG** enables you to machine holes using a single-lip deep hole drill. It is possible to enter a recessed starting point. You can define the direction of rotation and the rotational speed for moving into and retracting from the hole.

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 102, 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
- 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 INFEEED/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 102
- 9 If programmed, the tool moves to 2nd set-up clearance at **FMAX**

Please note while programming:**NOTICE****Danger of collision!**

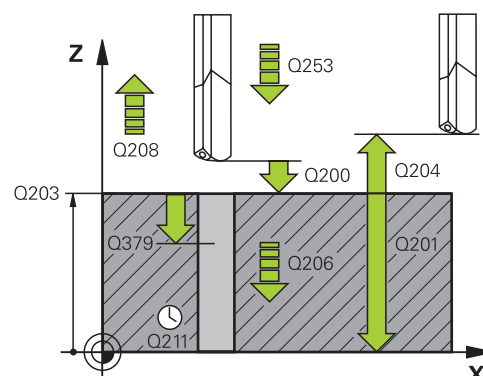
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ 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.
 - This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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**



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 INFED/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

- ▶ **Q426 Rot. dir. of entry/exit (3/4/5)?**: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input:
3: Turn the spindle with M3
4: Turn the spindle with M4
5: Move with stationary spindle
- ▶ **Q427 Spindle speed of entry/exit?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole.
Input range: 0 to 99999
- ▶ **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:
 $0.2 \times \mathbf{Q379}$; 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:
 $0.2 \times \mathbf{Q379} = 0.2 \times 2 = 0.4$; 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: **$0.8 \times Q379$** ; 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 position for chip removal is calculated as follows:
 $0.8 \times Q379 = 0.8 \times 2 = 1.6$; 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	$0.8 \cdot 2 = 1.6$	-0.4
2	5	0	2	$0.8 \cdot 5 = 4$	-3
2	10	0	2	$0.8 \cdot 10 = 8$ (Q200 =2, $8 > 2$, so the value 2 is used.)	-8
2	25	0	2	$0.8 \cdot 25 = 20$ (Q200 =2, $20 > 2$, so the value 2 is used.)	-23
2	100	0	2	$0.8 \cdot 100 = 80$ (Q200 =2, $80 > 2$, so the value 2 is used.)	-98
5	2	0	5	$0.8 \cdot 2 = 1.6$	-0.4
5	5	0	5	$0.8 \cdot 5 = 4$	-1
5	10	0	5	$0.8 \cdot 10 = 8$ (Q200 =5, $8 > 5$, so the value 5 is used.)	-5
5	25	0	5	$0.8 \cdot 25 = 20$ (Q200 =5, $20 > 5$, so the value 5 is used.)	-20
5	100	0	5	$0.8 \cdot 100 = 80$ (Q200 =5, $80 > 5$, so the value 5 is used.)	-95
20	2	0	20	$0.8 \cdot 2 = 1.6$	-1.6
20	5	0	20	$0.8 \cdot 5 = 4$	-4
20	10	0	20	$0.8 \cdot 10 = 8$	-8
20	25	0	20	$0.8 \cdot 25 = 20$	-20
20	100	0	20	$0.8 \cdot 100 = 80$ (Q200 =20, $80 > 20$, so the value 20 is used.)	-80

4.10 CENTERING (Cycle 240, DIN/ISO: G240, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Use Cycle **240 CENTERING** to machine center holes. You can specify the centering diameter or depth and an optional dwell time at the bottom.

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 setup clearance or to 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

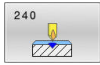
Danger of collision!

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!

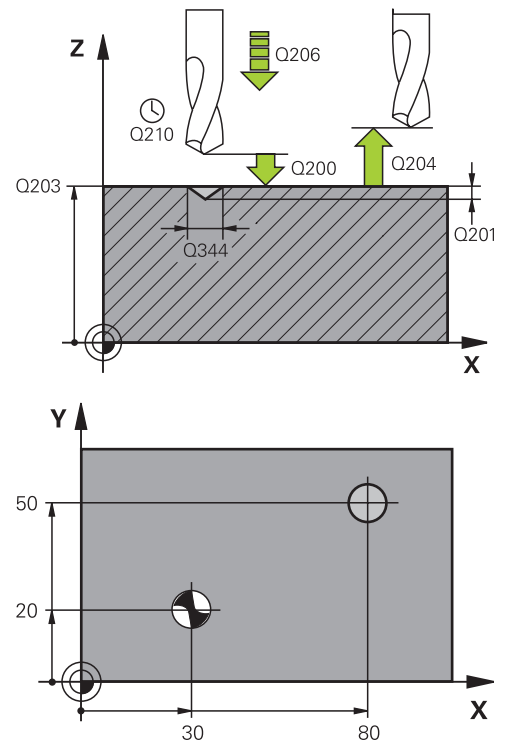
- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- Program a positioning block to position the tool at the starting point (hole center) in the working plane with 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.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

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 with reference to the active preset
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

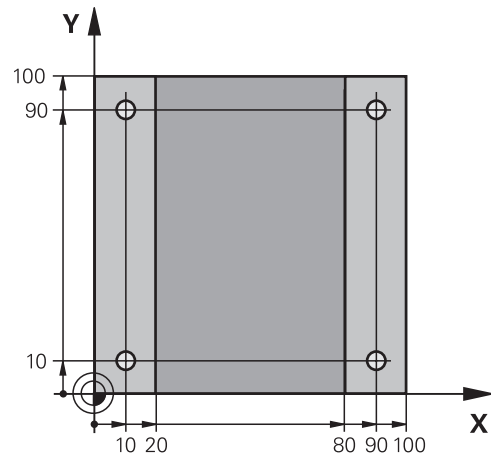


Example

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

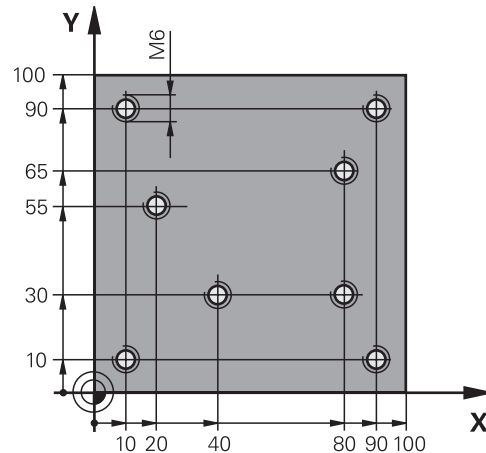
The drill hole coordinates are stored in the pattern definition PATTERN DEF POS and are called by the Control with CYCL CALL PAT.

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 112



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

**Cycles: Tapping /
thread milling**

5.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	TAPPING with a floating tap holder (Cycle 206, ISO: G206) <ul style="list-style-type: none"> ■ With a floating tap holder ■ Input of the dwell time at bottom 	113
	TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207) <ul style="list-style-type: none"> ■ Without a floating tap holder ■ Input of the dwell time at bottom 	116
	TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209, option 19) <ul style="list-style-type: none"> ■ Without a floating tap holder ■ Input of chip breaking behavior 	120
	THREAD MILLING (Cycle 262, DIN/ISO: G262, option 19) <ul style="list-style-type: none"> ■ Milling a thread into pre-drilled material 	126
	THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263, option 19) <ul style="list-style-type: none"> ■ Milling a thread into pre-drilled material ■ Machining a countersunk chamfer 	130
	THREAD MILLING (Cycle 264, DIN/ISO: G264, option 19) <ul style="list-style-type: none"> ■ Drilling into solid material ■ Milling a thread 	134
	HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265, option 19) <ul style="list-style-type: none"> ■ Milling a thread into solid material 	138
	OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267, option 19) <ul style="list-style-type: none"> ■ Milling an external thread ■ Machining a countersunk chamfer 	142

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

Application

The thread is cut in one or more passes. A floating tap holder is used.

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 set-up clearance at the end of the dwell time. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
- 4 At the set-up clearance, the direction of spindle rotation reverses once again.



Operating notes:

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

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.

Please note while programming:**NOTICE****Danger of collision!**

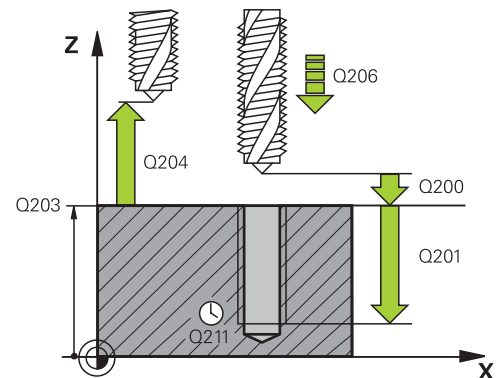
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ 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.
 - 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.
 - This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

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 with reference to the active preset
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	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q206=150	;FEED RATE FOR PLNGNG
Q211=0.25	;DWELL TIME AT DEPTH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

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

F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the **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)

Application



Refer to your machine manual!

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.

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

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 It then reverses the direction of spindle rotation and the tool is retracted to set-up clearance. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
- 4 The control stops the spindle turning at that set-up clearance



Operating notes:

- 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.

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)

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 **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 will stand still after the end of the cycle. In this case, 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.
 - This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

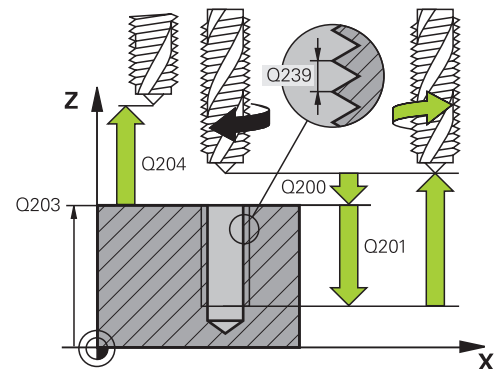


If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

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 with reference to the active preset
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	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q239=+1	;THREAD PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

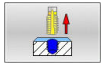
Retracting after a program interruption

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.



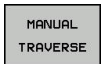
- ▶ Press **NC start**
- ▶ The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

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, option 19)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

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

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.

Cycle run

- 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 2nd set-up clearance at **FMAX**
- 6 The control stops the spindle turning at that set-up clearance



Operating notes:

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

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.

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 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 will stand still after the end of the cycle. In this case, 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.
 - This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

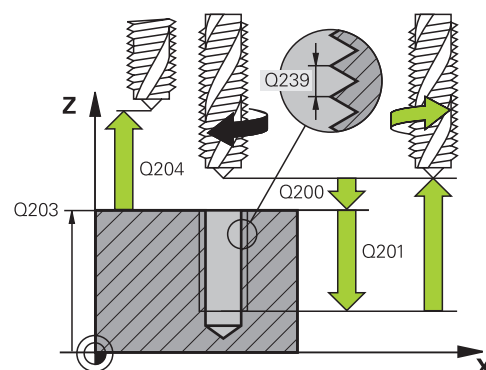


If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis can leave the acceleration path within this distance

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 with reference to the active preset
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 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 re-cut 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. Maximum increase to maximum speed of the active gear stage.
Input range: 0.0001 to 10.



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

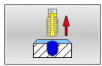
Retracting after a program interruption

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 anti-clockwise 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 want 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 MIRRORING MIRROR IMAGE** in only one axis.

5.6 THREAD MILLING (Cycle 262, DIN/ISO: G262, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With this cycle, you can mill a thread into pre-drilled material.

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 2nd setup clearance



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.

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 the thread milling cycle, the tool will make a compensation movement in the tool axis before the approach. The length of the compensation movement is at most half of the thread pitch. This can result in a collision.

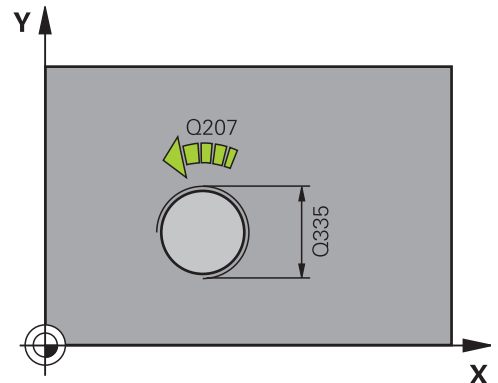
- ▶ Ensure sufficient space in the hole!

- 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 program the thread depth =0, the cycle will not be executed.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.

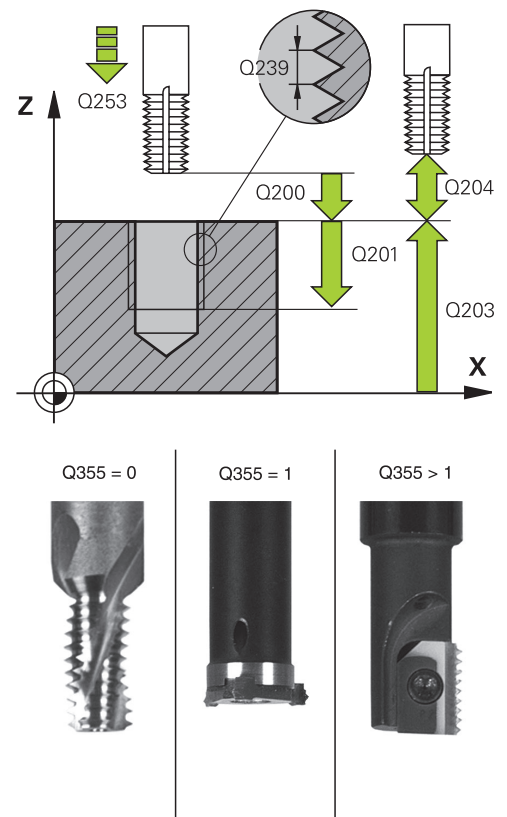
Cycle parameters



- ▶ **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 with reference to the active preset
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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With this cycle, you can mill a thread into pre-drilled material. In addition, you can use it to machine a countersunk chamfer.

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 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.

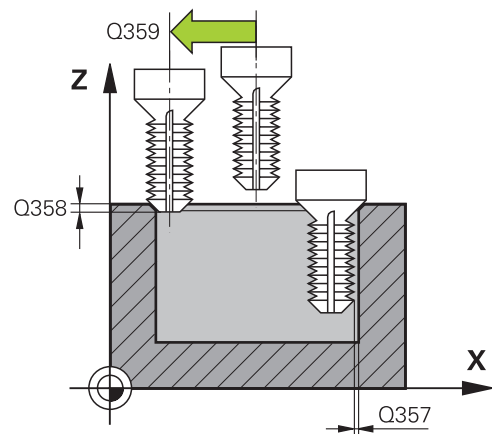
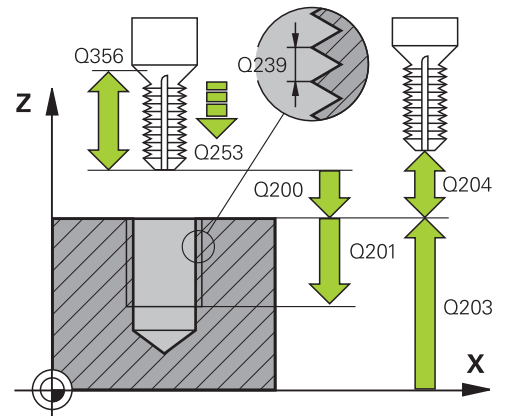
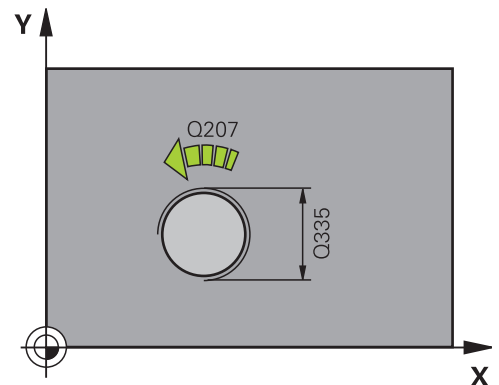


Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

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 with reference to the active preset
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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With this cycle, you can drill into solid material, machine a counterbore, and finally mill a thread.

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 retracted at rapid traverse to set-up clearance, and then moved again 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 drilling 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 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.

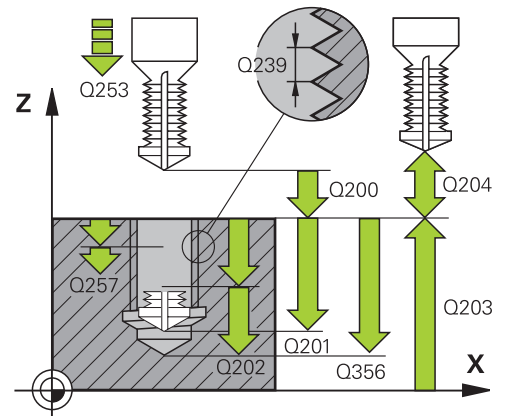
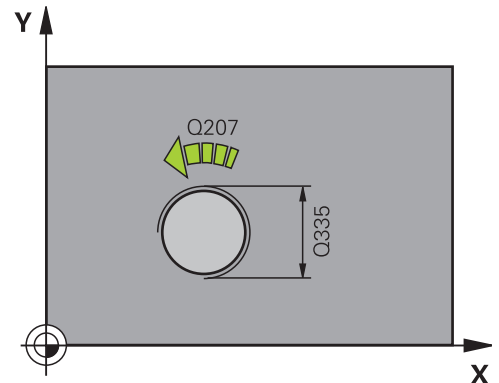


Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

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
Q202=5	;PLUNGING DEPTH
Q258=0.2	;UPPER ADV STOP DIST
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=0.2	;DIST FOR CHIP BRKNG

- ▶ **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 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
- ▶ **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 with reference to the active preset
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**

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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With this cycle, you can mill a thread into solid material. In addition, you can choose to machine a counterbore before or after milling the thread.

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 prepositioning 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 2nd setup clearance

Please note while programming:

NOTICE

Danger of collision!

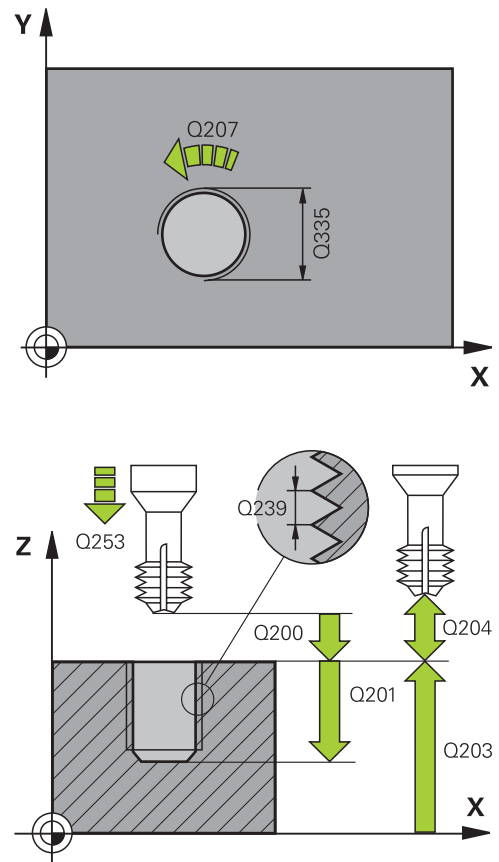
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ 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.

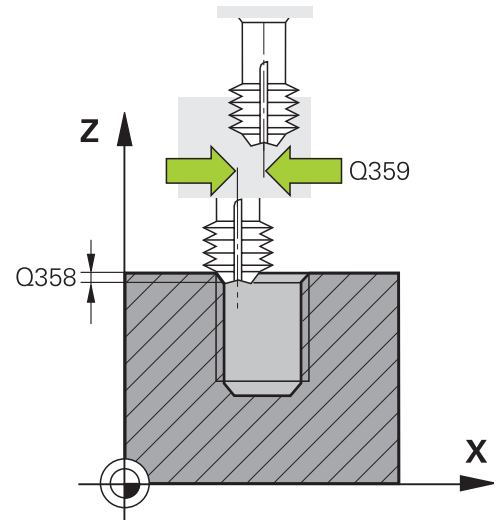
Cycle parameters



- ▶ **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 with reference to the active preset
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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With this cycle, you can mill an external thread. In addition, you can use it to machine a countersunk chamfer.

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 2nd setup clearance

Please note while programming:**NOTICE****Danger of collision!**

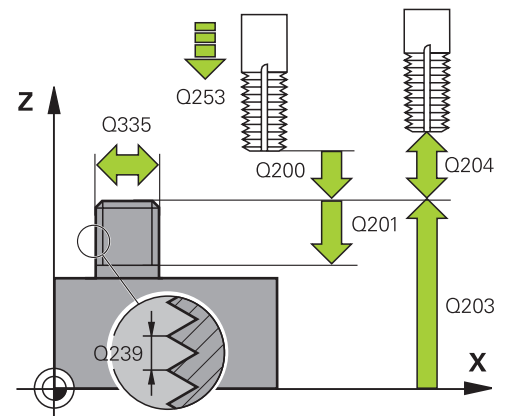
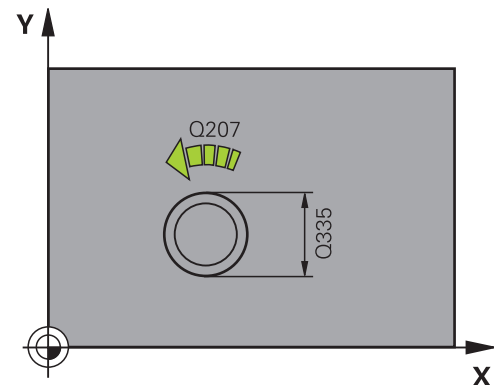
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ 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.
 - 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).

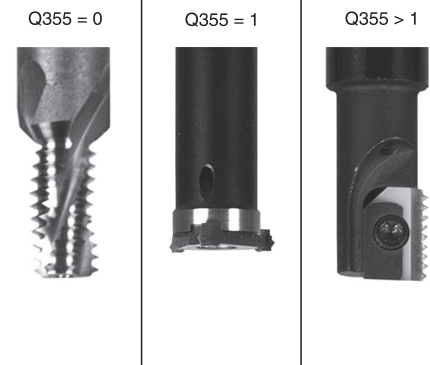
Cycle parameters



- ▶ **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 with reference to the active preset
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

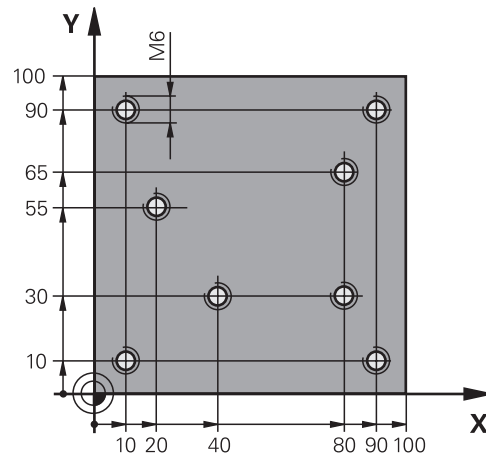
Example: Thread milling

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

**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
	RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical 	151
	CIRCULAR POCKET (Cycle 252, DIN/ISO: G252, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Plunging strategy: helical or vertical 	158
	SLOT MILLING (Cycle 253, DIN/ISO: G253, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical 	165
	CIRCULAR SLOT (Cycle 254, DIN/ISO: G254, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical 	169
	RECTANGULAR STUD (Cycle 256, DIN/ISO: G256, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Approach position: selectable 	175
	CIRCULAR STUD (Cycle 257, DIN/ISO: G257, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Input of the start angle ■ Helical infeed starting from the workpiece blank diameter 	180
	POLYGON STUD (Cycle 258, DIN/ISO: G258, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Helical infeed starting from the workpiece blank diameter 	184
	FACE MILLING (Cycle 233, DIN/ISO: G233, Option 19) <ul style="list-style-type: none"> ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls 	190

6.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251, Option 19)

Application

Use Cycle **251** 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

Cycle run

Roughing

- 1 The tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter **Q366**.
- 2 The control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowances (**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 set-up clearance above the current plunging depth. From there, the tool is returned 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.
- 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.

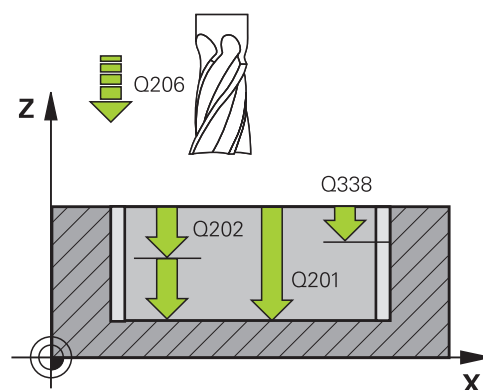
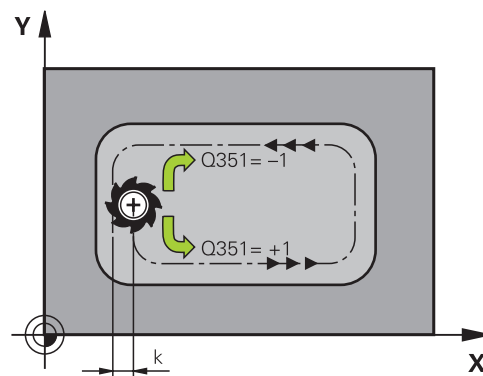
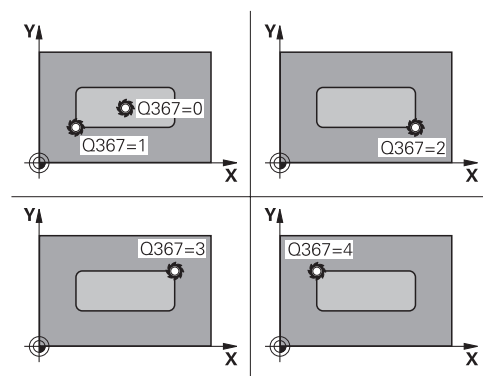
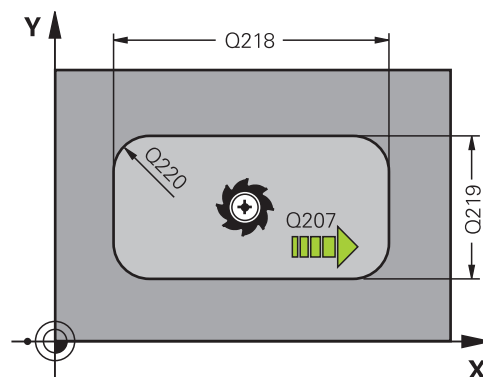
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed.
- Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- Cycle **251** takes the cutting width **RCUTS** from the tool table.

Further information: "Plunging strategy Q366 with RCUTS",
Page 157

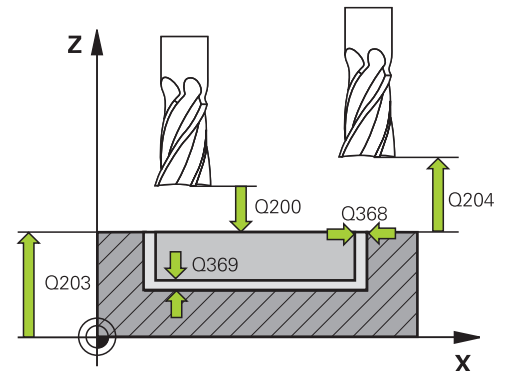
Cycle parameters



- ▶ **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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **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 with reference to the active preset
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**



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

- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:

0: vertical plunging. The control plunges the tool vertically, 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 will display an error message. If necessary, define the value of the cutting width **RCUTS** in the tool table

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. If necessary, define the value of the cutting width **RCUTS** in the tool table

PREDEF: The control uses the value from the GLOBAL DEF NC block

Further information: "Plunging strategy Q366 with RCUTS", Page 157

- **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

Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q370=1	;TOOL PATH OVERLAP
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

Plunging strategy Q366 with RCUTS

Helical plunging Q366 = 1

RCUTS > 0

- The control takes the cutting width **RCUTS** into account when calculating the helical path. The greater **RCUTS**, the smaller the helical path.
- Formula for calculating the helical radius:
$$\text{Helicalradius} = R_{\text{corr}} - \text{RCUTS}$$

 R_{corr} : Tool radius **R** + tool radius oversize **DR**
- If moving on a helical path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- The control does not monitor or modify the helical path.

Reciprocating plunge Q366 = 2

RCUTS > 0

- The control moves the tool along the complete reciprocating path.
- If moving on a reciprocating path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- The control moves the tool along one half of the reciprocating path.

6.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

Use Cycle **252** 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

Cycle run

Roughing

- 1 The control first moves the tool at rapid traverse to 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 allowances (**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 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 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 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

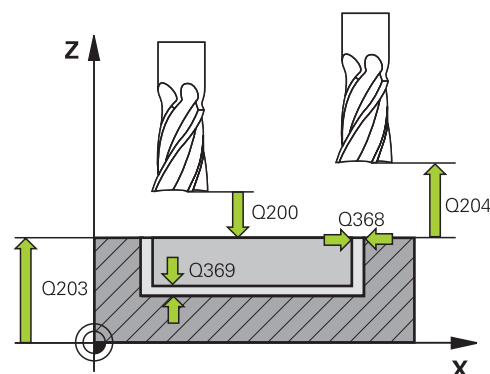
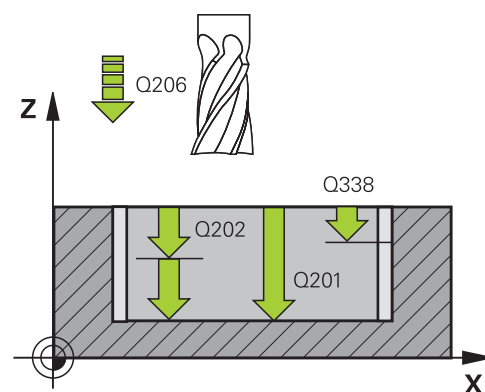
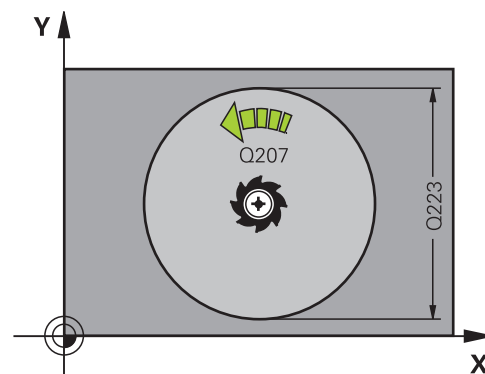
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the tool table is inactive, you must always plunge vertically (**Q366=0**) because you cannot define a plunging angle.
- Pre-position the tool in the machining plane to the starting position (circle center) with radius compensation **R0**.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- 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 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.
 - This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
 - Cycle **252** takes the cutting width **RCUTS** from the tool table.
- Further information:** "Plunging strategy Q366 with RCUTS", Page 164

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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **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 with reference to the active preset
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. If necessary, define the value of the **RCUTS** cutting width in the tool table
 Alternatively **PREDEF**
Further information: "Plunging strategy Q366 with RCUTS", Page 164
- ▶ **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

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	

Plunging strategy Q366 with RCUTS

Behavior with RCUTS

Helical plunging **Q366=1**:

RCUTS > 0

- The control takes the cutting width **RCUTS** into account when calculating the helical path. The greater **RCUTS**, the smaller the helical path.
- Formula for calculating the helical radius:

$$\text{Helicalradius} = R_{\text{corr}} - \text{RCUTS}$$

$$R_{\text{corr}}: \text{Tool radius } R + \text{tool radius oversize } DR$$
- If moving on a helical path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- **suppressPlungeErr=on** (no. 201006)
 If moving on a helical path is not possible due to limited space, the control will reduce the helical path.
- **suppressPlungeErr=off** (no. 201006)
 If moving on a helical radius is not possible due to limited space, the control will display an error message.

6.4 SLOT MILLING (Cycle 253, DIN/ISO: G253, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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

Cycle run

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 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 a finishing allowance has been defined during pre-machining, the control first finishes the slot walls, using 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!

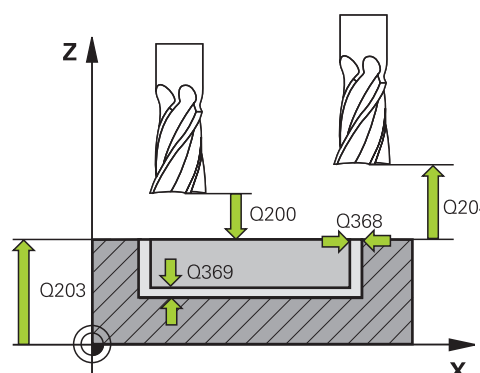
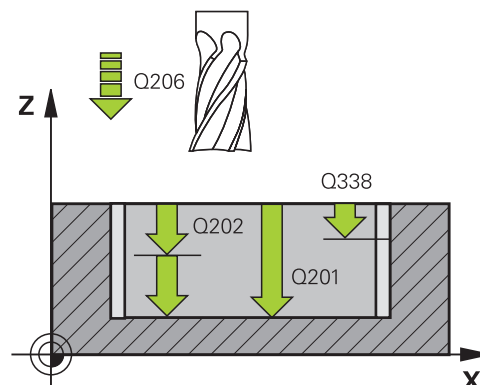
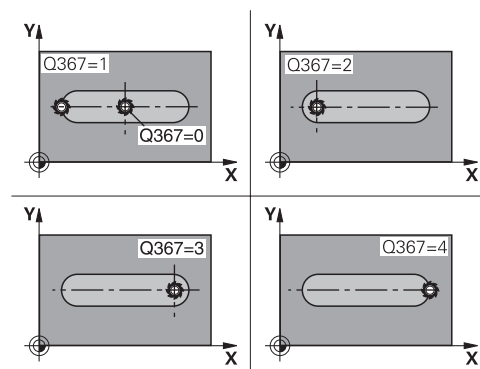
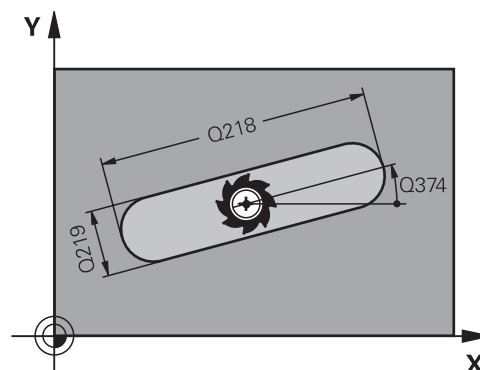
- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- 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).
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

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 with respect to the tool position when the cycle is called:
0: Tool position = center of slot
1: Tool position = left end of slot
2: Tool position = center of left slot 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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **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
- ▶ **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 with reference to the active preset
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

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
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

6.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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

Cycle run

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 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

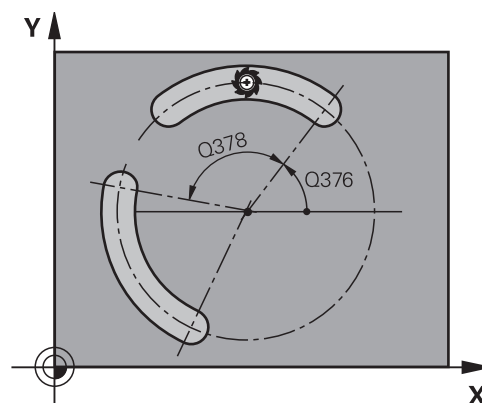
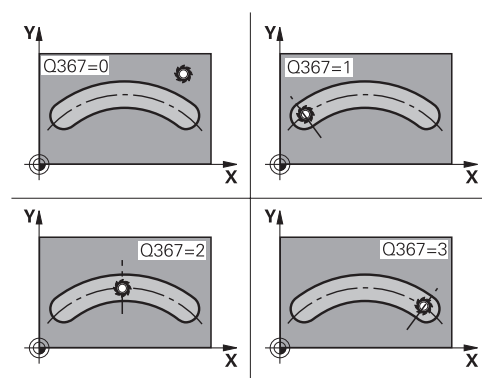
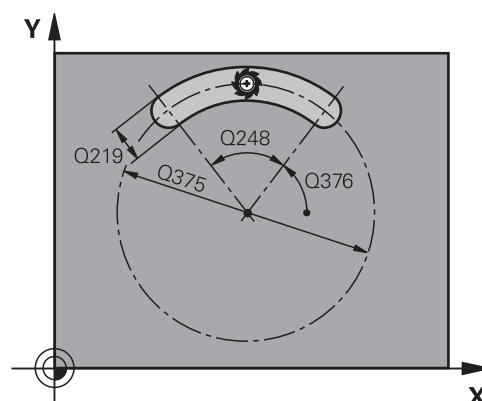
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the tool table is inactive, you must always plunge vertically (**Q366=0**) because you cannot define a plunging angle.
- Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- 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.
- Slot position 0 is not allowed if you use Cycle **254** 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

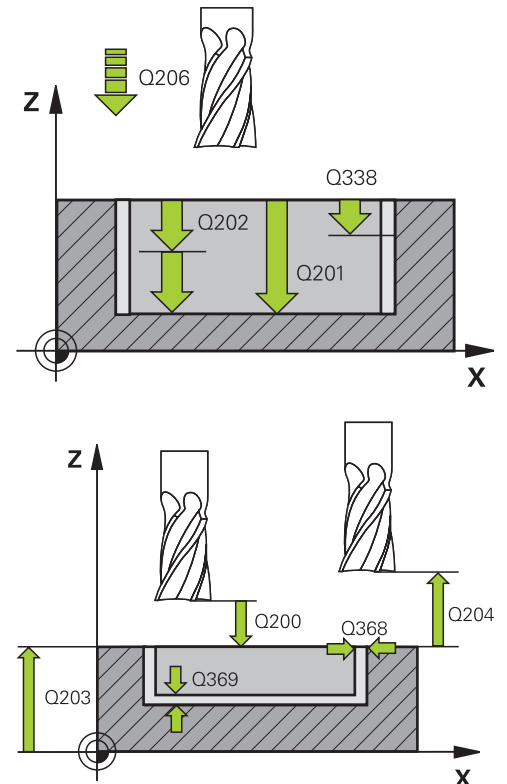
Cycle parameters



- ▶ **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 with respect 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 direction of spindle rotation is taken into account:
 +1 = climb milling
 -1 = up-cut milling
PREDEF: The control uses the value from a **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
Q207=500	;FEED RATE MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH

- ▶ **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 with reference to the active preset
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

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, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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.

Cycle run

- 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 default position (**Q437=0**) is 2 mm to the right of the stud blank
- 2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to 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.

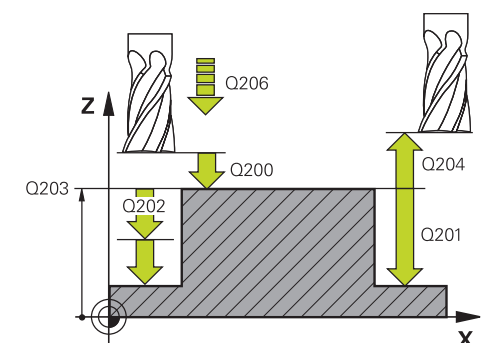
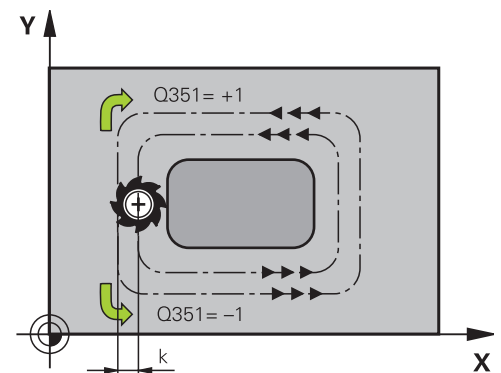
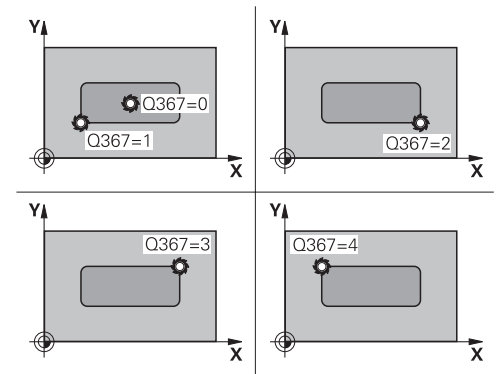
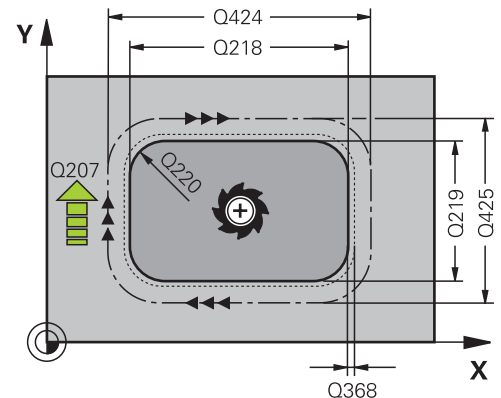
- ▶ Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement
- ▶ Leave room next to the stud for the approach motion
- ▶ At least tool diameter + 2 mm
- ▶ At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer.
Input range: -99999.9999 to +99999.9999
- ▶ **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 with respect 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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **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 with reference to the active preset
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	;INFEEED 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 finishingSide 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, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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.

Cycle run

- 1 If the current position of the tool is below the 2nd set-up clearance, the control then lifts it off and retracts it 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 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 firsts departs on a tangential path and is then retracted in the tool axis to the 2nd set-up clearance 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!**

There is a danger of collision if there is insufficient room next to the stud.

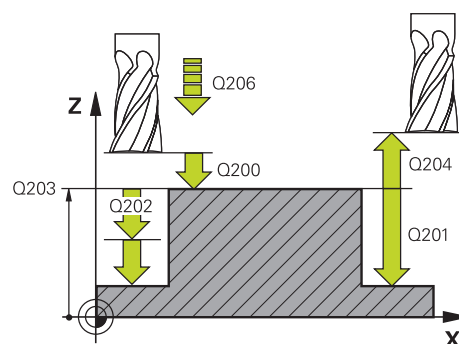
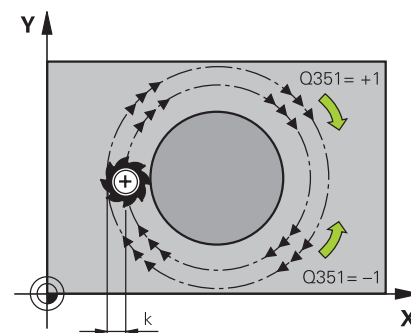
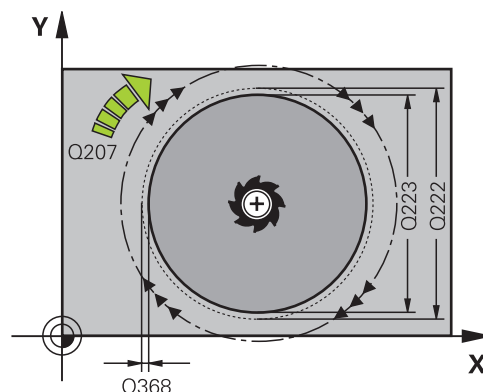
- ▶ In this cycle, the control performs an approach movement
- ▶ To define the exact starting position, enter a starting angle between 0° and 360° in parameter **Q376**
- ▶ Depending on the starting angle **Q376**, the following amount of space must be left next to the stud: At least tool diameter +2 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.
- 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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 direction of spindle rotation is taken into account:
 +1 = climb milling
 -1 = up-cut milling
PREDEF: The control uses the value from a **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 with reference to the active preset
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	;INFEEED 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, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

Use Cycle **258** to machine a regular polygon by machining the contour outside. The milling operation is carried out on a spiral path based on the diameter of the workpiece blank.

Cycle run

- 1 If, at the beginning of machining, the work piece is positioned below the 2nd set-up clearance, the control will retract the tool back to 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 2nd set-up 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 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 $+360^\circ$
- ▶ 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

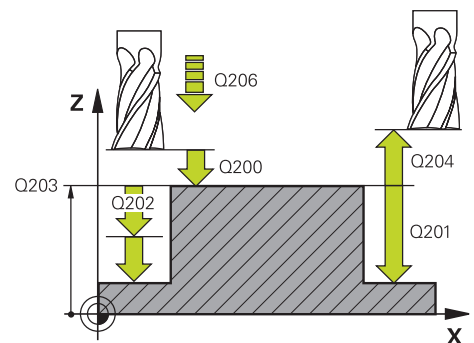
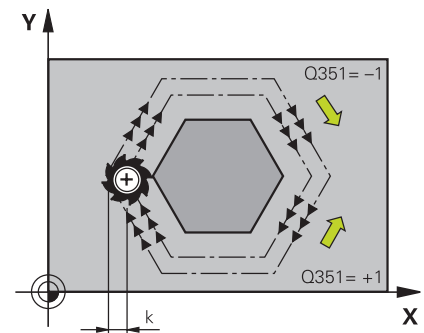
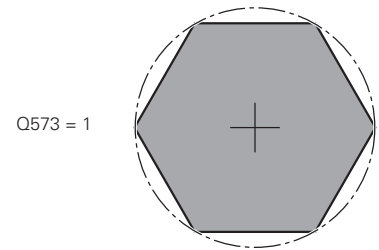
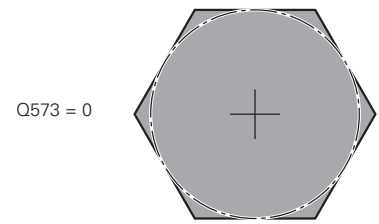
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the start of the cycle you will have to pre-position the tool in the working plane. In order to do so, move the tool with radius compensation **R0** to the center of the stud.
- The control automatically pre-positions 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Cycle parameters



- ▶ **Q573 Inscr.circle/circumcircle (0/1)?**: Definition of whether the dimension **Q571** shall reference the inscribed circle or the perimeter:
0= dimension 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 entered here 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 $+360^\circ$
- ▶ **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer.
 Input range: -99999.9999 to +99999.9999
- ▶ **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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **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 with reference to the active preset
Input range: -99999.9999 to 99999.9999

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	;INFEEED FOR FINISHING
Q385=500	;FINISHING FEED RATE
9 L X+50 Y+50 R0 FMAX M3 M99	

- ▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.
Input range: 0 to 99999.9999; alternatively **PREDEF**
- ▶ **Q370 Path overlap factor?:** $Q370 \times \text{tool radius} = \text{stepover factor } k$.
Input range: 0.0001 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, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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

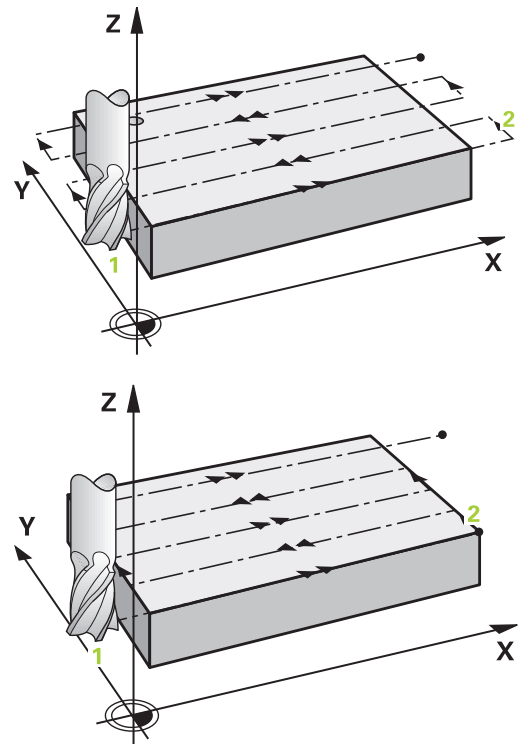
Cycle run

- 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 set-up clearance in the spindle axis
- 3 The tool then moves in the spindle 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.

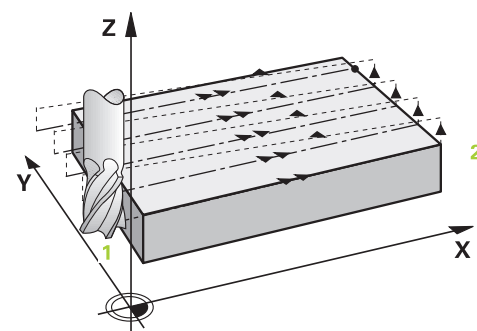
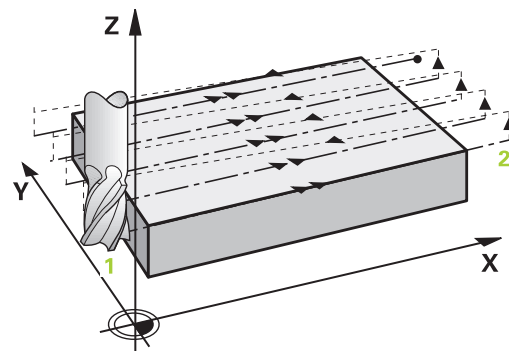
- 4 The control moves the tool to end point **2** at the programmed feed rate for milling
- 5 Then the control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor 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 machined completely.
- 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 spindle 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

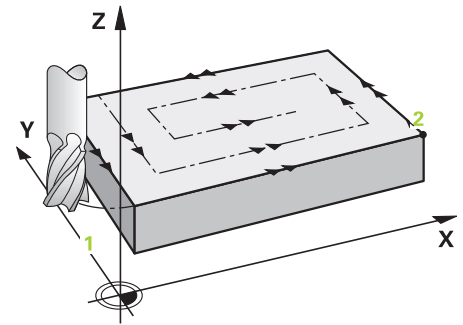
The strategies **Q389=2** and **Q389=3** differ in the overtravel during face milling. If **Q389=2**, the end point lies outside of the surface, with **Q389=3**, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the 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**.

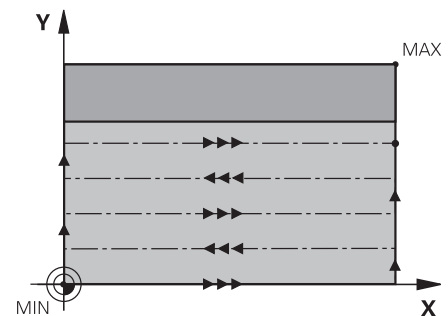


Strategy Q389=4

- 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 working plane to the starting position with radius compensation R0. Note the machining direction.
 - The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
 - 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.
 - This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

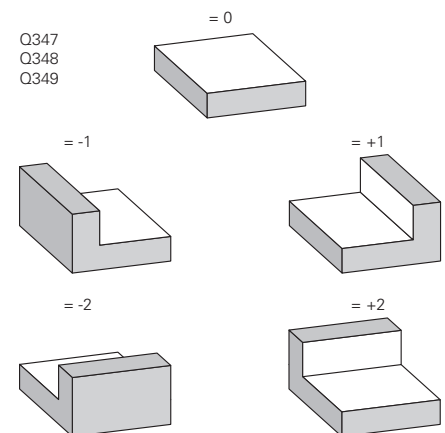
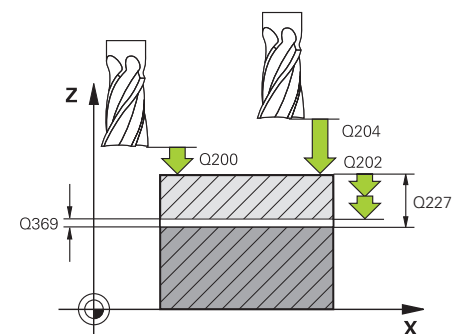
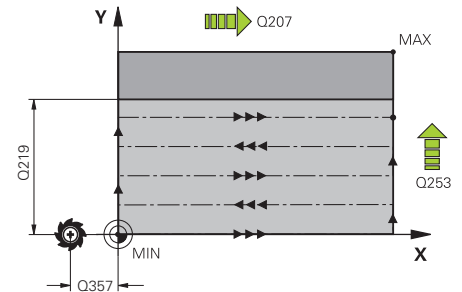


Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

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**

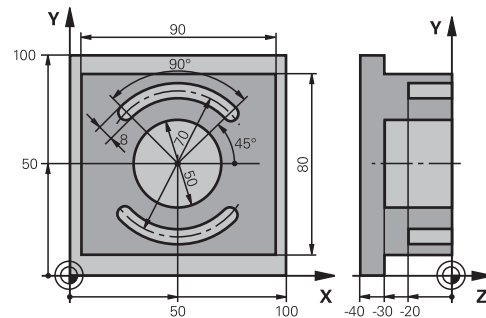
Example

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 plane 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	;INFEEED FOR FINISHING	
Q200=2	;SET-UP CLEARANCE	
Q203=+0	;SURFACE COORDINATE	
Q204=50	;2ND SET-UP CLEARANCE	
Q370=1	;TOOL PATH OVERLAP	
Q366=1	;PLUNGE	
Q385=750	;FINISHING FEED RATE	
Q439=0	;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99		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	;INFEEED FOR FINISHING	
Q200=2	;SET-UP CLEARANCE	
Q203=+0	;SURFACE COORDINATE	
Q204=50	;2ND SET-UP CLEARANCE	
Q366=1	;PLUNGE	
Q385=500	;FINISHING FEED RATE	
Q439=0	;FEED RATE REFERENCE	
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
	DATUM SHIFT (Cycle 7, DIN/ISO: G54) <ul style="list-style-type: none"> ■ Shifting contours directly in the NC program ■ Or shifting contours using datum tables 	203
	MIRRORING (Cycle 8, DIN/ISO: G28) <ul style="list-style-type: none"> ■ Mirroring contours 	210
	ROTATION (Cycle 10, DIN/ISO: G73) <ul style="list-style-type: none"> ■ Rotating contours in the working plane 	211
	SCALING (Cycle 11, DIN/ISO: G72) <ul style="list-style-type: none"> ■ Resizing contours 	213
	AXIS-SPECIFIC SCALING (Cycle 26) <ul style="list-style-type: none"> ■ Axis-specific resizing of contours 	214
	WORKING PLANE (Cycle 19, DIN/ISO: G80, Option 8) <ul style="list-style-type: none"> ■ Executing machining operations in a tilted coordinate system ■ On machines with swivel heads and/or rotary tables 	216
	PRESETTING (Cycle 247, DIN/ISO: G247) <ul style="list-style-type: none"> ■ Datum setting during program run 	223

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)

Application

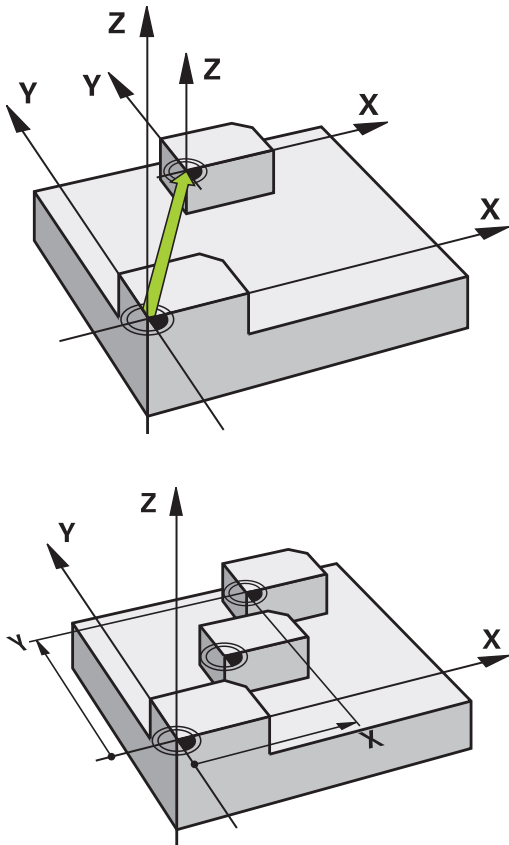
 Refer to your machine manual!

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.

Reset

- 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** 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 6 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

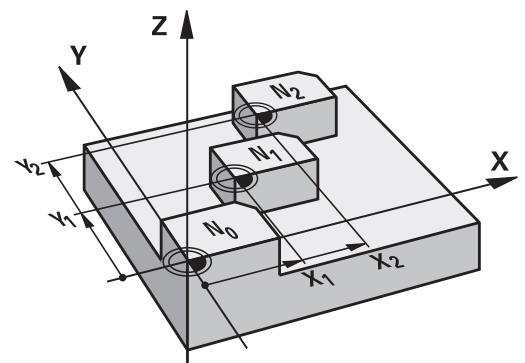
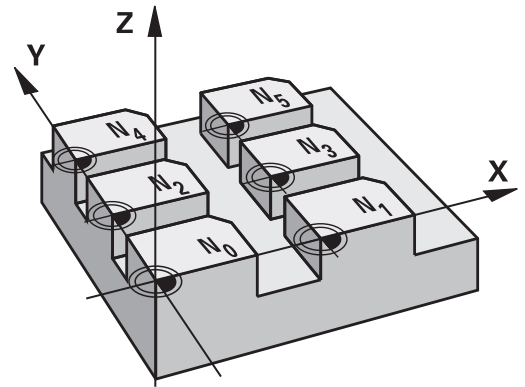
7.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

Application

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

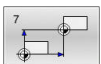
In the additional status display, the following data from the datum table is shown:

- 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** 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:

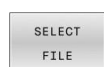


- ▶ Press the **PGM CALL** key



- ▶ Press the **SELECT DATUM TABLE** soft key
- ▶ Enter the complete path name of the datum table

or



- ▶ Press the **SELECT FILE** soft key
- ▶ Confirm your input with the **END** key.



Programming and operating notes:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
- 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:



- ▶ Press the **PGM MGT** key



- ▶ Press the **SELECT TYPE** soft key





















- ▶ 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

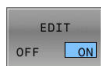


- ▶ Press the **SELECT COMPENS. TABLES** soft key



- ▶ 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.

A datum change 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.

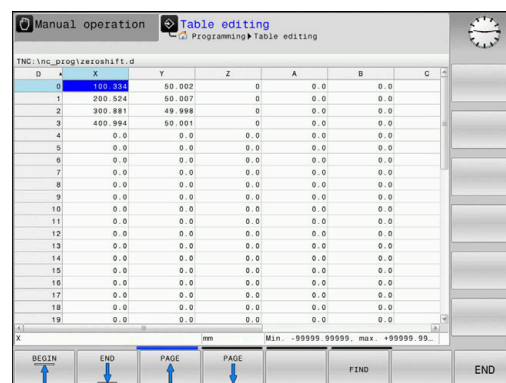
Further information: User's Manual for Conversational Programming

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)

Application

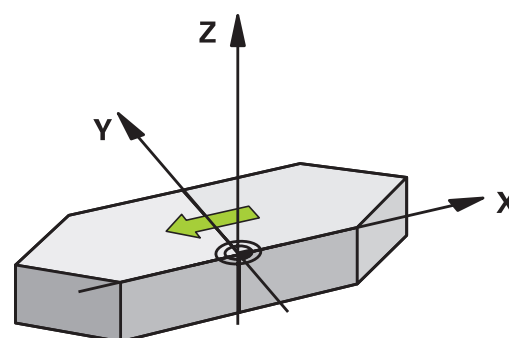
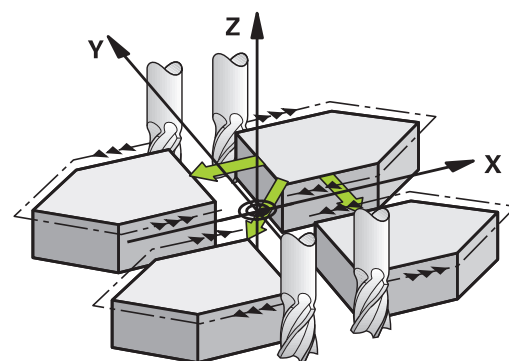
The control can machine the mirror image of a contour in the working plane.

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.



Reset

Program Cycle **8 MIRRORING** again with **NO ENT**.

Please note while programming:

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



For working 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)

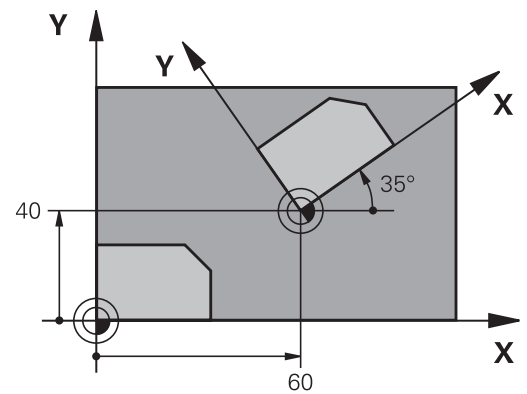
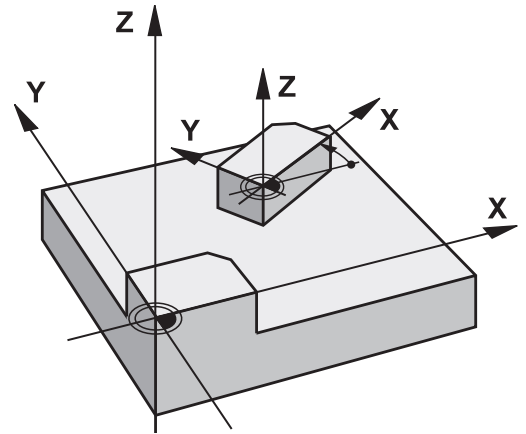
Application

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

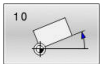


Reset

Program Cycle **10 ROTATION** again and specify 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**, move both axes of the working plane to activate the 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)

Application

The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The scaling factor 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)

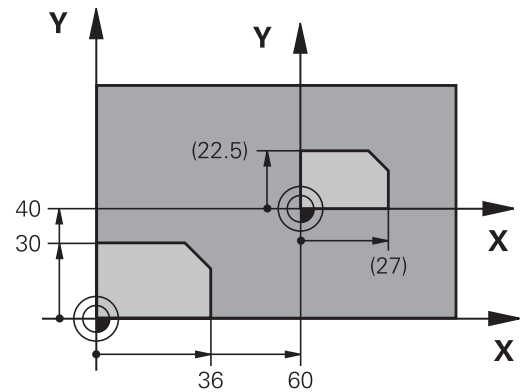
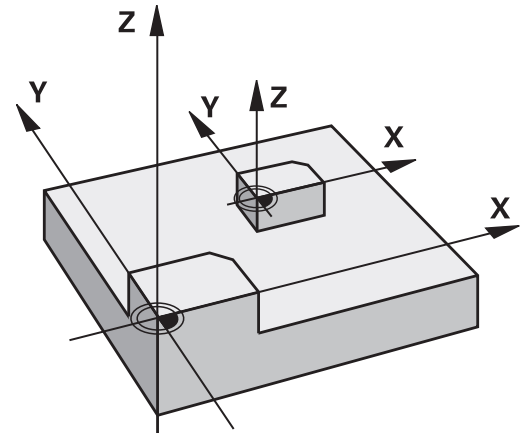
Reduction: SCL less than 1 (down to 0.000 001)



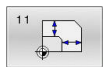
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Reset

Program Cycle **11 SCALING FACTOR** again and specify 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 FACTOR
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1
  
```

7.7 AXIS-SPECIFIC SCALING (Cycle 26)

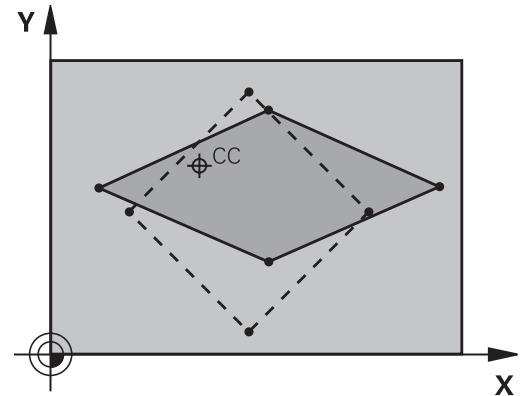
Application

Use Cycle **26** to account for shrinkage and allowance factors for each axis.

The scaling factor 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.

Reset

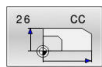
Program Cycle **11 SCALING FACTOR** again and enter a scaling factor of 1 for the corresponding axis.



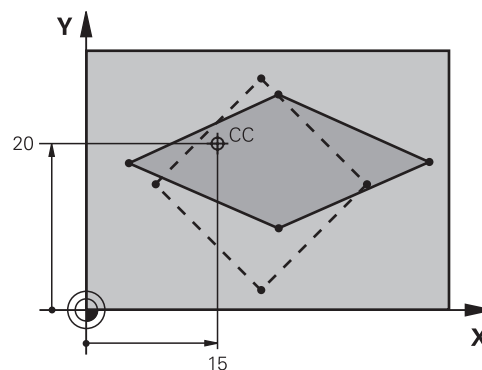
Please note while programming:

- 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 contour is enlarged or reduced relative to the center, and not necessarily (as in Cycle **11 SCALING FACTOR**) relative 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 8)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Use Cycle **19** to 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.



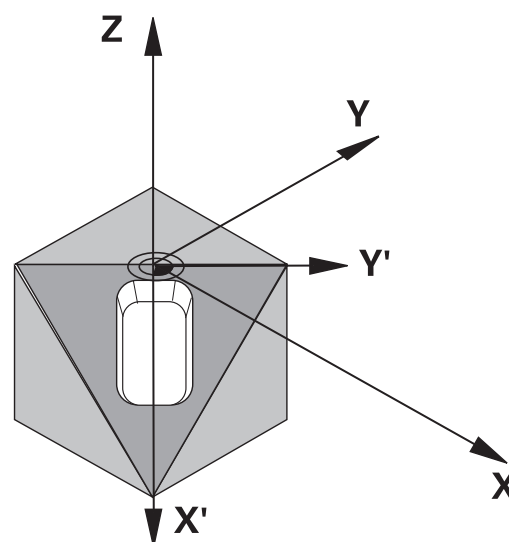
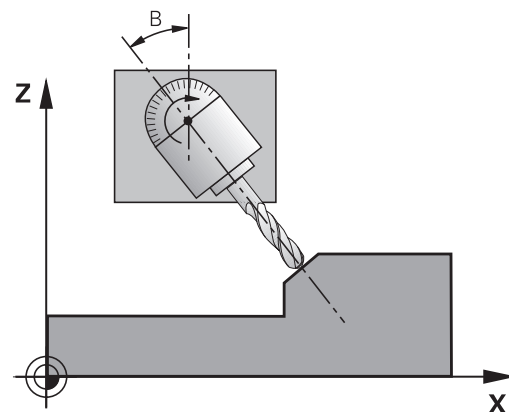
Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

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** function to **Active** in Manual Operation mode, the angular value entered in this menu will be overwritten by Cycle **19 WORKING PLANE**.



Please note while programming:

The machine tool builder specifies whether the programmed angles are interpreted by the control 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 be executed in the **FUNCTION MODE MILL** machining mode.
- In combination with a radial facing slide kinematics, this cycle can also be used in the **FUNCTION MODE TURN** machining mode.
- 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.



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.

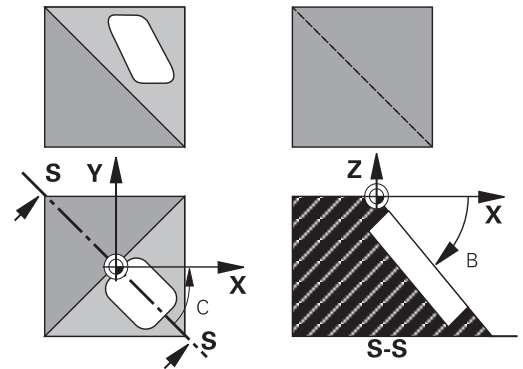
Cycle parameters



- **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 Cycle **19 WORKING PLANE**. Enter an angular value of 0° for all rotary axes. Then, redefine Cycle **19 WORKING PLANE**. 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 need to 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. For using spatial angles, 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 the use of functions such as **M94** (modulo rotary axes) in order to prevent discrepancies between actual and nominal positions of the rotary axes for multiple calls.

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 (**NOML** and **ACTL**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. This means that the position 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 miscellaneous function **M130**, you can move the tool, while the coordinate system is tilted, to positions that reference the non-tilted coordinate system.

With a tilted working plane, it is also possible to position the axes using straight-line blocks that reference the machine coordinate system (NC blocks with **M91** or **M92**). Constraints:

- 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 the activation of Cycle **19**, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

- 1 Activate datum shift
- 2 Activate **Tilt working plane**
- 3 Activate rotation
- ...
- Workpiece machining
- ...
- 1 Reset the rotation
- 2 Reset **Tilt working plane**
- 3 Reset the 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 different angles, if necessary
- ▶ Reset Cycle **19** by programming 0° for all rotary axes
- ▶ Redefine Cycle **19** in order to deactivate the working plane
- ▶ 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 Programming of Measuring Cycles for Workpieces and Tools

Further information: User's Manual for Setup, Testing and Running NC Programs

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

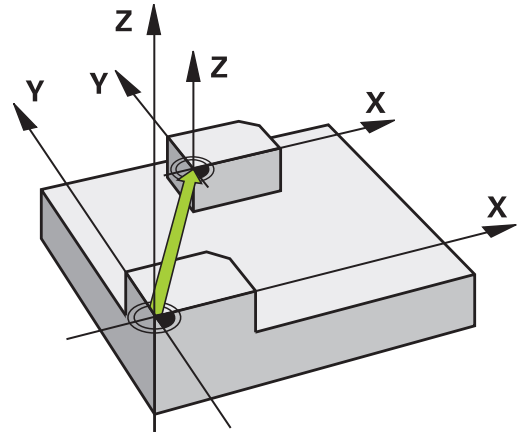
Application

Use Cycle **247 PRESETTING** to activate a preset defined in the preset table as the new preset.

After cycle definition, all coordinate input and datum shifts (absolute or incremental) reference 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** 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

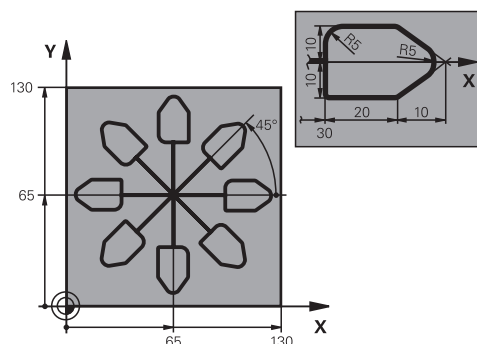
In the additional status display (**STATUS POS.**), the control shows the active preset number behind the **Preset** dialog.

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

Cycles:
Pattern definitions

8.1 Fundamentals

Overview

The control provides three cycles for machining point patterns:

Soft key	Cycle	Page
	POLAR PATTERN (Cycle 220, DIN/ISO: G220, Option 19) <ul style="list-style-type: none"> ■ Defining a circular pattern ■ Full circle or pitch circle ■ Input of start and end angles 	230
	CARTESIAN PATTERN (Cycle 221, DIN/ISO: G221, Option 19) <ul style="list-style-type: none"> ■ Defining a linear pattern ■ Input of an angle of rotation 	233
	DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224, Option 19) <ul style="list-style-type: none"> ■ Converting text to a DataMatrix code to be used as a point pattern ■ Input of position and size 	236

You can combine the following cycles with Cycles **220**, **221**, and **224**:

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 the following cycles only with Cycles **220** and **221**:

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 MILLING/CNTSNKG
Cycle 264	THREAD DRILLING/MILLING
Cycle 265	HEL. THREAD DRILLING/MILLING
Cycle 267	OUTSIDE THREAD MILLING



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 64

Further information: "Pattern definition with PATTERN DEF",
Page 57

8.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

This cycle enables you to define a point pattern as a full or pitch circle. It can be used for a previously defined machining cycle.

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 2nd set-up clearance (spindle axis)
 - Approach the starting point in the working plane
 - Move to 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 line or a circular arc. 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 have been completed

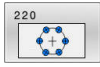


If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

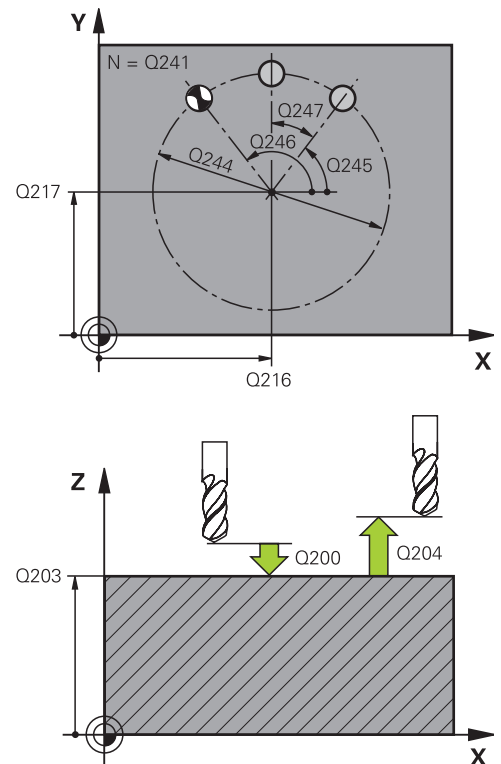
Please note while programming:

- Cycle **220** is DEFactive. In addition, Cycle **220** automatically calls the last defined machining cycle.
- If you combine one of the machining cycles **200** to **209** and **251** to **267** with Cycle **220** or Cycle **221**, the set-up clearance, the workpiece surface, and the 2nd set-up clearance from Cycle **220** or **221** are 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).

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 with reference to the active preset
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

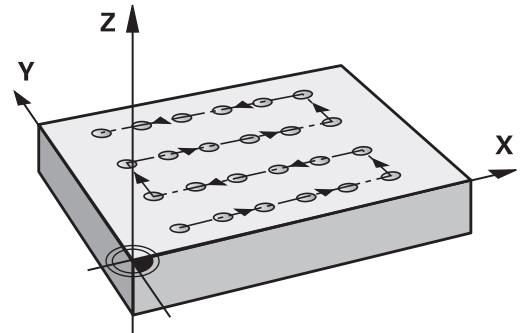
8.3 CARTESIAN PATTERN (Cycle 221, DIN/ISO: G221, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

This cycle enables you to define a point pattern as lines. It can be used for a previously defined machining cycle.



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 2nd set-up clearance (spindle axis)
 - Approach the starting point in the working plane
 - Move to 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 procedure (step 6) will be repeated until all machining operations of the second line have been completed
- 8 The tool then moves to the starting point of the next line
- 9 All subsequent lines are machined in a reciprocating movement.



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

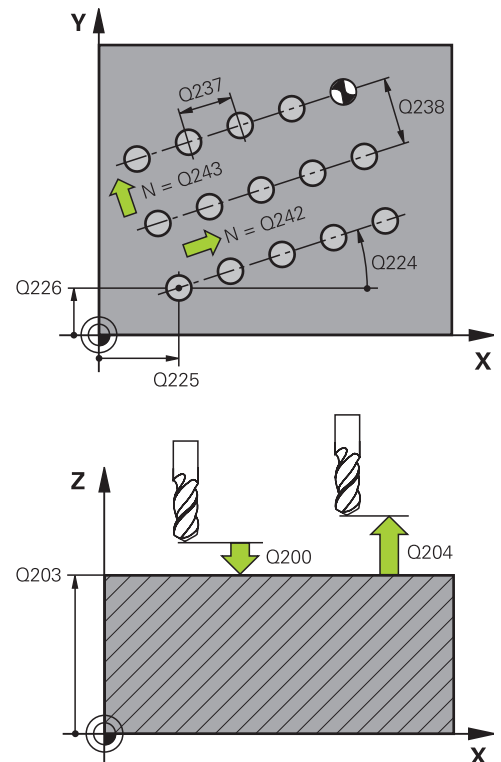
Please note while programming:

- Cycle **221** is DEFactive. In addition, Cycle **221** automatically calls the last defined machining cycle.
- If you combine one of the machining cycles **200** to **209** and **251** to **267** with Cycle **221**, the set-up clearance, the workpiece surface, the 2nd safety clearance, and the rotary position from Cycle **221** are effective.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

Cycle parameters



- ▶ **Q225 Starting point in 1st axis?** (absolute):
Coordinate of the starting point in the principal axis of the working plane.
Input range: -99999.9999 to +99999.9999
- ▶ **Q226 Starting point in 2nd axis?** (absolute):
Coordinate of the starting point in the secondary axis of the working plane.
Input range: -99999.9999 to +99999.9999
- ▶ **Q237 Spacing in 1st axis?** (incremental): Spacing between the individual points on the line.
Input range: -99999.9999 to +99999.9999
- ▶ **Q238 Spacing in 2nd axis?** (incremental): Spacing between the individual lines.
Input range: -99999.9999 to +99999.9999
- ▶ **Q242 Number of columns?**: Number of machining operations on a line (row).
Input range: 0 to 99999
- ▶ **Q243 Number of lines?**: The number of lines.
Input range: 0 to 99999
- ▶ **Q224 Angle of rotation?** (absolute): Angle by which the entire pattern is rotated. The center of rotation is located at the starting point.
Input range: -360 to +360
- ▶ **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 with reference to the active preset
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, Option 19)

Application

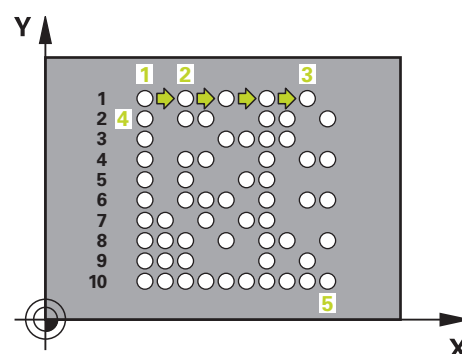


This function must be enabled and adapted by the machine tool builder.

Use Cycle **224 DATAMATRIX CODE PATTERN** to convert text to a so-called DataMatrix code. This code will be used as a point pattern for a previously defined fixed cycle.

Cycle run

- 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 2nd set-up clearance (spindle axis)
 - Approach the starting point in the working 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 machining cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle **224** will be effective for the selected machining cycle.

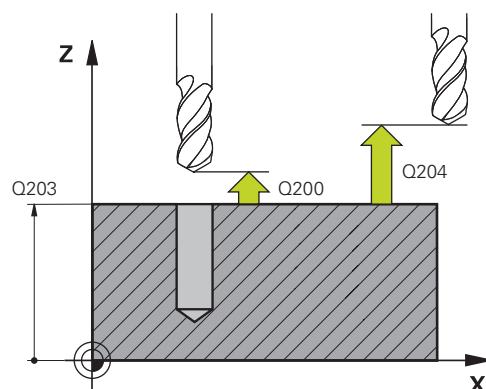
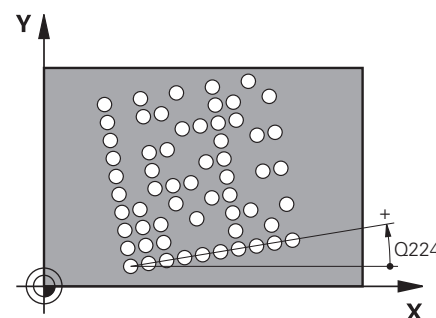
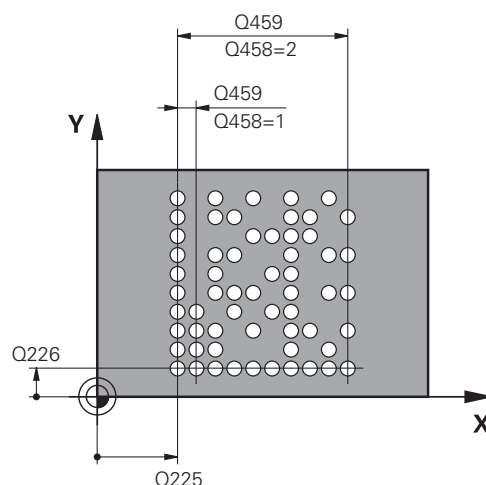
- ▶ Check the machining sequence using a graphic simulation
- ▶ Carefully test the NC program or program section in **Program run, single block** operating mode

- 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 machining cycle.

Cycle parameters



- ▶ **Q225 Starting point in 1st axis?** (absolute):
Coordinate in the lower left corner of the code in the principal axis.
Input range: -99999.9999 to +99999.9999
- ▶ **Q226 Starting point in 2nd axis?** (absolute):
Definition of a coordinate in the lower left corner of the code in the secondary axis.
Input range: -99999.9999 to +99999.9999
- ▶ **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.
Input range: -360 to +360
- ▶ **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 with reference to the active preset
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

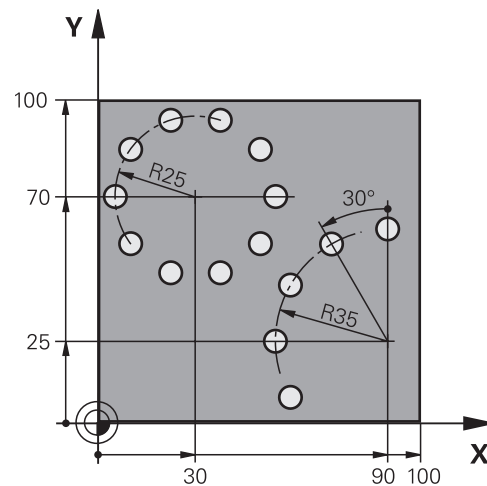


Example

54 CYCL DEF 224 DATAMATRIX CODE PATTERN	
Q225=+0	;STARTNG PNT 1ST AXIS
Q226=+0	;STARTNG PNT 2ND AXIS
QS501=""	;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

**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 list of subcontours (subprogram numbers) you specified in Cycle **14 CONTOUR**.



Programming and operating notes:

- 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 AUSRAEUMEN ...
19 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING ...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING ...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM







```

Cycle properties





- 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, allowances and set-up clearance, can be entered centrally in Cycle **20 CONTOUR DATA**.

Overview

Soft key	Cycle	Page
	CONTOUR (Cycle 14, DIN/ISO: G37) ■ Listing the contour subprograms	245
	CONTOUR DATA (Cycle 20, DIN/ISO: G120, option 19) ■ Input of machining information	250
	PILOT DRILLING (Cycle 21, DIN/ISO: G121, option 19) ■ Machining a hole for non-center cutting tools	252
	ROUGHING (Cycle 22, DIN/ISO: G122, option 19) ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	254
	FLOOR FINISHING (Cycle 23, DIN/ISO: G123, option 19) ■ Finishing with finishing allowance for the floor from Cycle 20	258
	SIDE FINISHING (Cycle 24, DIN/ISO: G124, option 19) ■ Finishing with side finishing allowance from Cycle 20	260

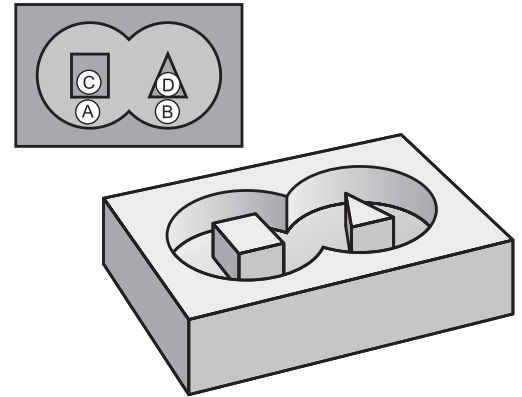
Enhanced cycles:

Soft key	Cycle	Page
	CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270, option 19) ■ Input of contour data for Cycle 25 or 276	263
	CONTOUR TRAIN (Cycle 25, DIN/ISO: G125, option 19) ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	265
	TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275, Option 19) ■ Machining of open and closed contours using trochoidal milling.	269
	THREE-D CONT. TRAIN (Cycle 276, DIN/ISO: G276, option 19) ■ Machining of open and closed contours ■ Detection of residual material ■ 3-D contours—additional processing of coordinates from the tool axis	274

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

Application

In Cycle **14 CONTOUR**, list all subprograms that are to be superimposed to define the overall contour .



Please note while programming!

- 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

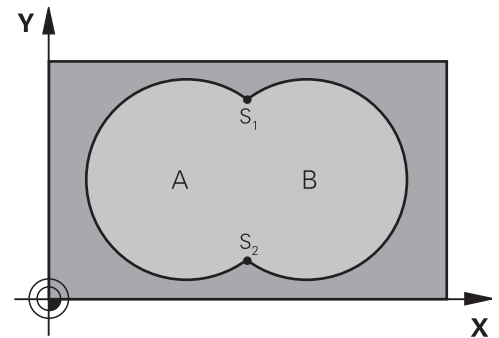


- **Label numbers for the contour:** Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm each input with the ENT key. Confirm each input with the **END** key. Input of up to 12 subprogram numbers: 1 to 65 535

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
```

```
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
```

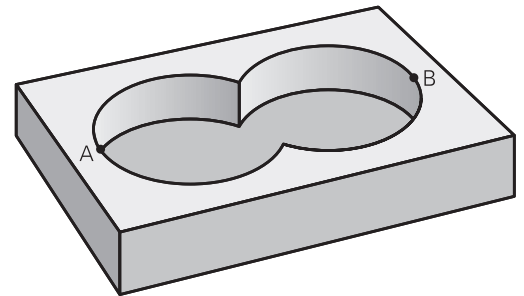
Subprogram 2: Pocket B

```
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

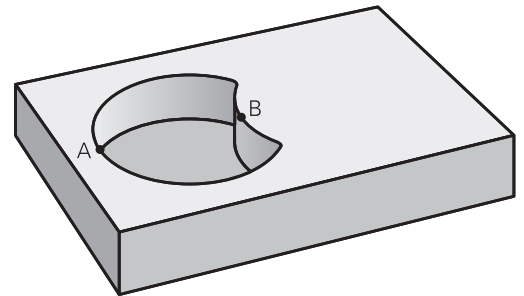
Surface B:

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

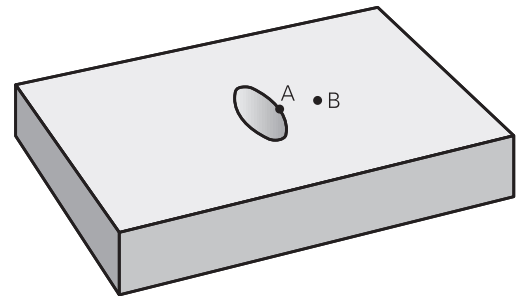
59 C X+40 Y+50 DR-

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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

Use Cycle **20** to specify machining data for the subprograms describing the subcontours.

Please note while programming!

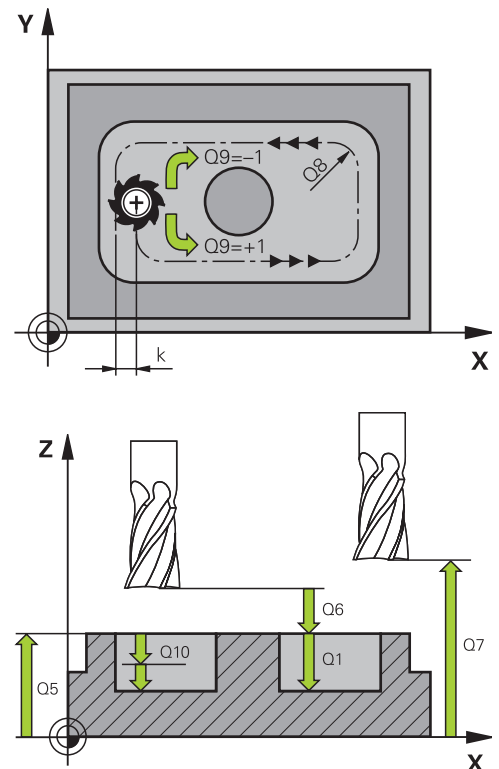
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **20** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **20** are valid for Cycles **21** to **24**.
- The algebraic sign for the 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



- ▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and bottom of pocket.
Input range: -99999.9999 to 99999.9999
- ▶ **Q2 Path overlap factor?:** $Q2 \times \text{tool radius} = \text{stepover factor } k$.
Input range: +0.0001 to 1.9999
- ▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the 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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

Use Cycle **21 PILOT DRILLING** if you machine a contour and then use a tool for roughing it out which has no center-cut end mill (ISO 1641). 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 finishing allowance for side and the finishing 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 programming the call of Cycle **21** you need to program two further cycles:

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

Cycle run

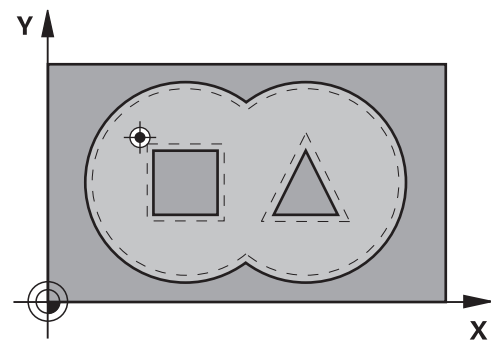
- 1 The tool first positions the tool in the plane (the position results from the contour that you previously defined with Cycle **14** or **SEL CONTOUR**, and from the information on the rough-out tool)
- 2 The tool then moves at rapid traverse **FMAX** to set-up clearance. (specify the set-up clearance in Cycle **20 CONTOUR DATA**)
- 3 The tool drills from the current position to the first plunging depth at the programmed feed rate **F**.
- 4 Then, the tool retracts at rapid traverse **FMAX** to the starting position and advances again to the first plunging depth minus the advanced stop distance **t**
- 5 The advanced stop distance is automatically calculated by the control:
 - At a total hole depth up to 30 mm: $t = 0.6 \text{ mm}$
 - At a total hole depth exceeding 30 mm: $t = \text{hole depth} / 50$
 - Maximum advanced stop distance: 7 mm
- 6 The tool then advances with another infeed at the programmed feed rate **F**.
- 7 The 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, option 19)

Application

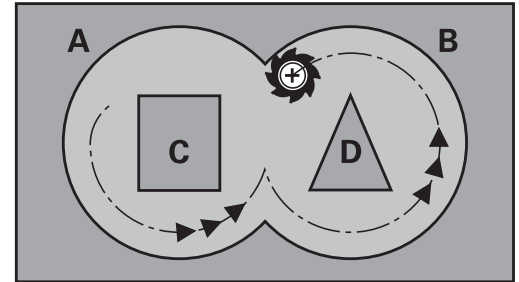


This function must be enabled and adapted by the machine tool builder.

Use Cycle **22 ROUGH-OUT** to define the technology data for roughing.

Before programming the call of Cycle **22**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable



Cycle run

- 1 The control positions the tool above the cutter infeed point, taking the finishing 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 The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B)
- 4 In the next step, the 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.
 - 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.
 - This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.
 - 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 the geometry 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**))



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

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 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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With Cycle **23 FLOOR FINISHING**, you can finish your contour by taking the finishing allowance for the floor into account that has been 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 programming the call of Cycle **23**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable **PILOT DRILLING**
- Cycle **22 ROUGH-OUT**, if necessary **ROUGH-OUT**

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.

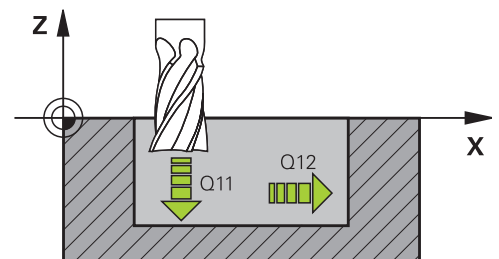
Please note while programming:**NOTICE****Danger of collision!**

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.
 - This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.

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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

With Cycle **24 SIDE FINISHING**, you can finish your contour by taking the finishing allowance for side into account that has been programmed in Cycle **20**. You can run this cycle in climb or up-cut milling.

Before programming the call of Cycle **24**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable **PILOT DRILLING**
- Cycle **22 ROUGH-OUT**, if necessary **ROUGH-OUT**

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.



Operating note:

- 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.

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.

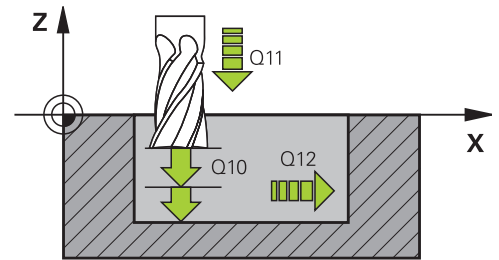
- ▶ After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- ▶ Make sure to program an absolute position after the cycle, no incremental traverse

- 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 was defined in Cycle **20**, the control generates 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 applies if you machine with 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**.
- If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.

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, option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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

Please note while programming:

- 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, option 19)

Application

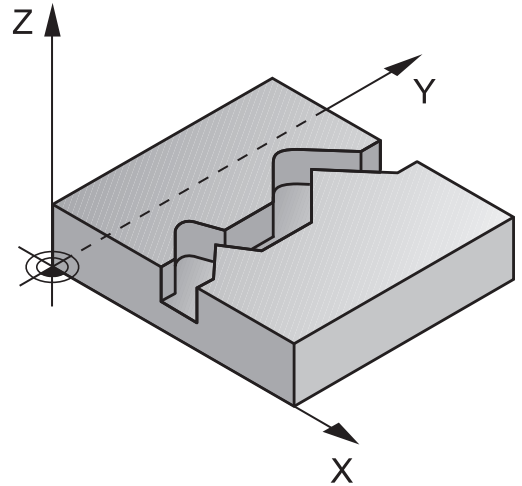


This function must be enabled and adapted by the machine tool builder.

In conjunction with Cycle **14 CONTOUR**, this cycle enables you to machine 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 be 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.

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, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

In conjunction with Cycle **14 KONTUR**, this cycle enables you to completely machine 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.

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

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

Program structure: Machining with SL cycles

```

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

```

Cycle run**Roughing closed slots**

In case of a closed slot, the contour description must always start with a straight-line block (**L** block).

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

Roughing 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 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!**

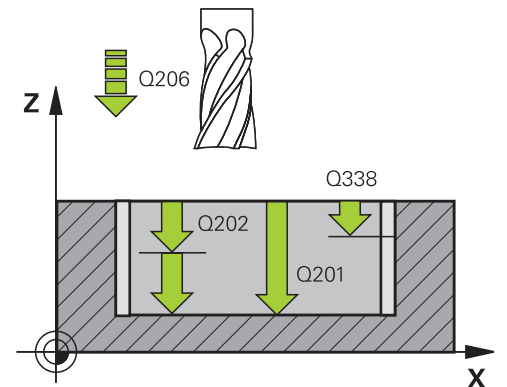
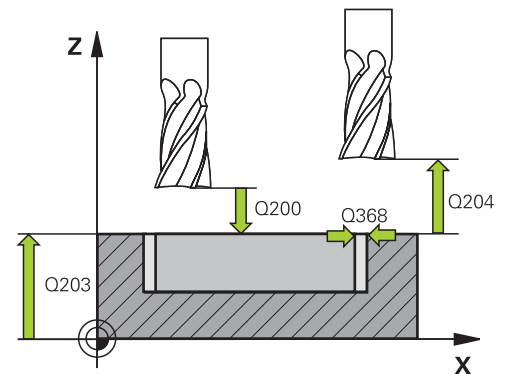
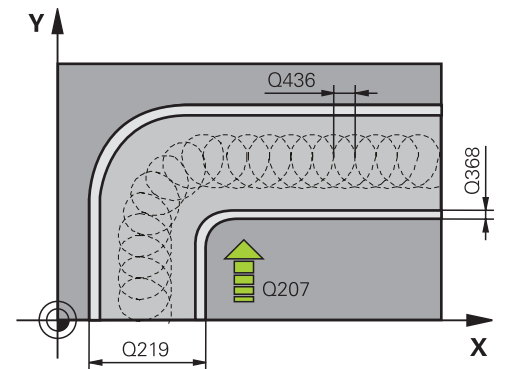
If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- ▶ 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.
 - If using Cycle **275 TROCHOIDAL SLOT**, you may 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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **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 with reference to the active preset
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**
- ▶ **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

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	

9.12 THREE-D CONT. TRAIN (Cycle 276, DIN/ISO: G276, option 19)

Application

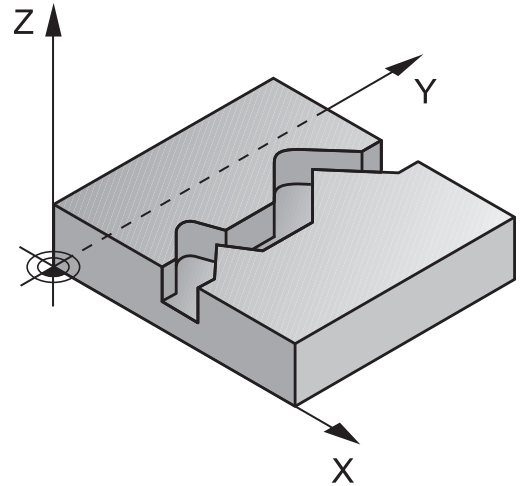


This function must be enabled and adapted by the machine tool builder.

In conjunction with Cycle **14 CONTOUR** and Cycle **270 CONTOUR TRAIN DATA**, this cycle enables you to machine 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**.



Cycle run

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. the 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. the 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 using Cycle **25 CONTOUR TRAIN**, you can define only one subprogram in Cycle **14 CONTOUR**.
- 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

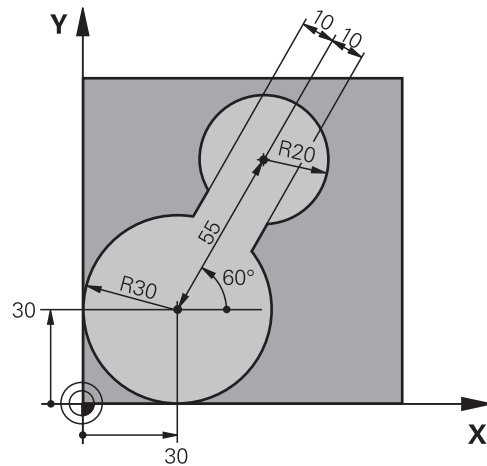
Example

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

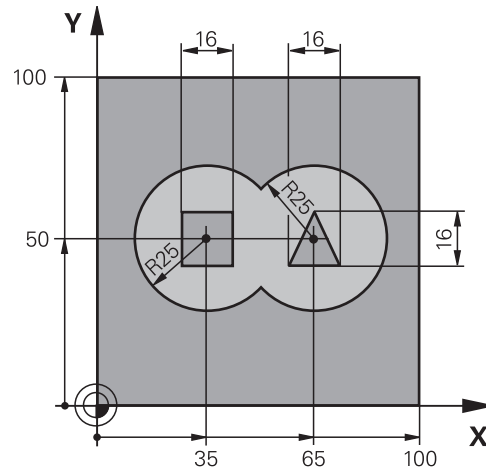
Example: Roughing-out and fine-roughing a pocket



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	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	

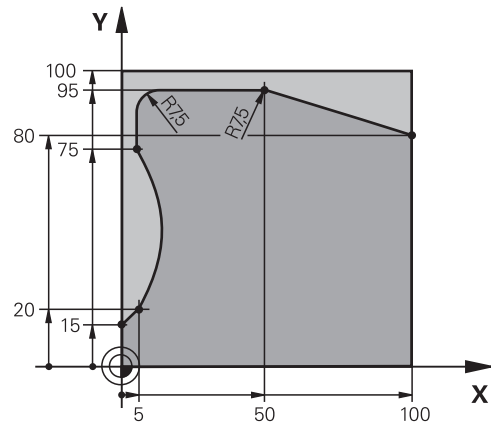
Example: Pilot drilling, roughing-out and finishing overlapping contours



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	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	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

**Cycles: Optimized
contour milling**

10.1 OCM cycles (option 167)

OCM fundamentals

General information



Refer to your machine manual!
Your machine manufacturer enables this function.

Using OCM cycles (**Optimized Contour Milling**), you can combine subcontours to form complex contours. These cycles provide more functionality than Cycles **22** to **24**. The OCM cycles feature 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



Programming and operating notes:

- 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 1.99; this corresponds to an angle of nearly 180°.

Contour

Specify the contour with **CONTOUR DEF / SEL CONTOUR** or with the OCM shape cycles **127x**.

Closed pockets can also be defined in Cycle **14**.

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **271 OCM CONTOUR DATA** or in the **127x** shape cycles.

CONTOUR DEF / SEL CONTOUR:

In **CONTOUR DEF / SEL CONTOUR**, the first contour can 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 Cycle **271 OCM CONTOUR DATA**
- ▶ 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.
- ▶ Define Cycle **272 OCM ROUGHING**



Programming notes:

- Subsequently defined contours that are outside the first contour will not be considered.
- The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth. Therefore, start programming the subcontour with the deepest pocket.

OCM shape cycles:

The shape defined in an OCM shape cycles can be a pocket, an island, or a boundary. Use the Cycles **128x** for programming an island or an open pocket.

Proceed as follows:

- ▶ Program a shape using cycles **127x**
- ▶ If the first shape will be an island or an open pocket, make sure to program boundary cycle **128x**.
- ▶ Define Cycle **272 OCM ROUGHING**

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
	OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167) <ul style="list-style-type: none"> ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block 	290
	OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167) <ul style="list-style-type: none"> ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable 	292
	OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167) <ul style="list-style-type: none"> ■ Finishing with finishing allowance for the floor from Cycle 271 ■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances) 	304
	OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167) <ul style="list-style-type: none"> ■ Finishing with side finishing allowance from Cycle 271 	307
	OCM CHAMFERING (Cycle 277, DIN/ISO: G277, Option 167) <ul style="list-style-type: none"> ■ Deburr the edges ■ Consider adjacent contours and walls 	309

OCM standard shapes:

Soft key	Cycle	Page
	OCM RECTANGLE (Cycle 1271, DIN/ISO: G1271, Option 167) <ul style="list-style-type: none"> ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners 	313
	OCM CIRCLE (Cycle 1272, DIN/ISO: G1272, Option 167) <ul style="list-style-type: none"> ■ Definition of a circle ■ Input of the circle diameter 	316
	OCM SLOT / RIDGE (Cycle 1273, DIN/ISO: G1273, Option 167) <ul style="list-style-type: none"> ■ Definition of a groove or ridge ■ Input of the width and the length 	318
	OCM POLYGON (Cycle 1278, DIN/ISO: G1278, Option 167) <ul style="list-style-type: none"> ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners 	320
	OCM RECTANGLE BOUNDARY (Cycle 1281, DIN/ISO: G1281, Option 167) <ul style="list-style-type: none"> ■ Definition of a bounding rectangle 	323
	OCM CIRCLE BOUNDARY (Cycle 1282, DIN/ISO: G1282, Option 167) <ul style="list-style-type: none"> ■ Definition of a bounding circle 	325

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

Application

Use Cycle **271 OCM CONTOUR DATA** to program machining data for the contour or the subprograms describing the subcontours. In addition, Cycle **271** enables you to define an open boundary for a pocket.

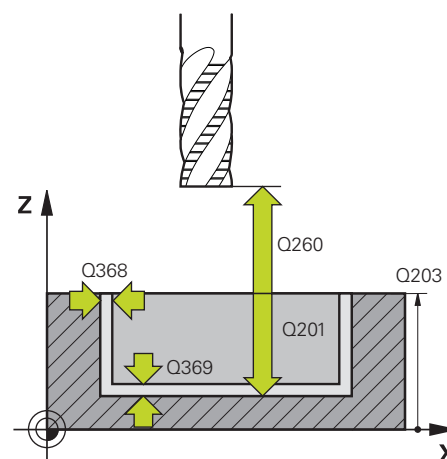
Please note while programming!

- 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**.

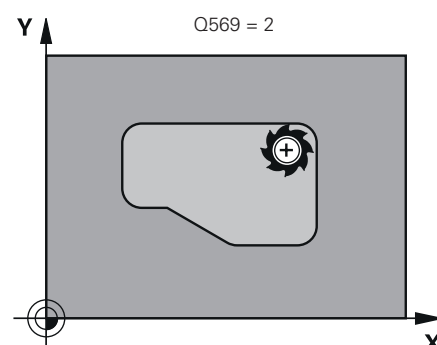
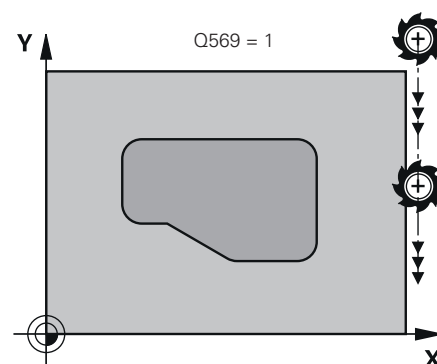
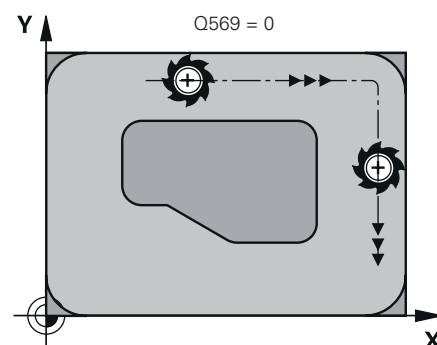
Cycle parameters



- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface with reference to the active preset
Input range: -99999.9999 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between the workpiece surface and the 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): Coordinate in the tool axis 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. The following contour must be an island
 - 2:** The first contour in **CONTOUR DEF** is interpreted as a bounding block. The following contour must be a pocket



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)

Application

Use Cycle **272 OCM ROUGHING** to define the technology data for roughing.

In addition, you can use the **OCM** cutting data calculator. The calculated cutting data help to achieve high material removal rates and therefore increase the productivity.

Further information: "OCM cutting data calculator (Option 167)", Page 296

Requirements

Before programming the call of Cycle **272**, you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**

Cycle run

- 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 or reciprocating movement to reach the first plunging depth. The finishing allowance for the side is taken into account

Further information: "Plunging behavior with Q569=0", Page 293

 - If you program **Q569=1**, the tool plunges outside the open boundary. The first plunging depth depends on the **Q575** plunging strategy
- 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 tool is moved to the next plunging depth and repeats the roughing procedure until the programmed contour is achieved
- 5 Finally, the tool retracts in the tool axis to the clearance height.

Plunging behavior with Q569=0

The control generally tries plunging with a helical path. If this is not possible, it tries plunging with a reciprocation movement.

The plunging behavior depends on:

- **Q207 FEED RATE MILLING**
- **Q568 PLUNGING FACTOR**
- **Q575 INFEEED STRATEGY**
- **ANGLE**
- **RCUTS**
- **R_{corr}** (tool radius **R** + tool oversize **DR**)

Helical:

The helical path is calculated as follows:

$$\text{Helicalradius} = R_{\text{corr}} - \text{RCUTS}$$

At the end of the plunging movement, the tool executes a semi-circular movement to provide sufficient space for the resulting chips.

Reciprocating

The reciprocation movement is calculated as follows:

$$L = 2 * (R_{\text{corr}} - \text{RCUTS})$$

At the end of the plunging movement, the tool executes a linear movement to provide sufficient space for the resulting chips.

Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- **CONTOUR DEF / SEL CONTOUR** will reset the tool radius that was used last. If you run this machining cycle with **Q438=-1** after **CONTOUR DEF / SEL CONTOUR**, the control assumes that no pre-machining has taken place yet.
- If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.
- If the path overlap factor **Q370** is less than 1, the plunging speed factor **Q579** must also be less than 1.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

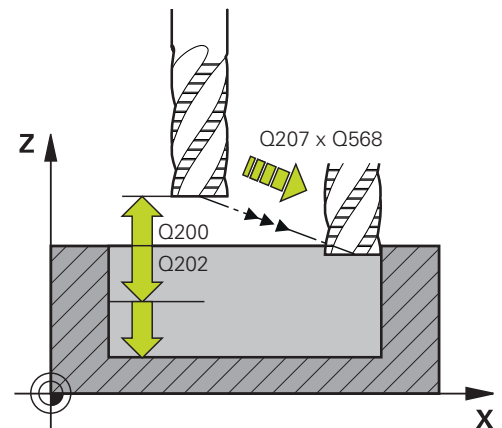


If required, use a center-cut end mill (ISO 1641).

Cycle parameters



- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut.
Input range: 0 to 99999.999
 - ▶ **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: 0.04 to 1.99; 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 in mm/min. This feed rate will be used below the coordinate surface, but outside the defined material.
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.
 - 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.
- Input range for number: -1 to 32767.9



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
Q576=+0 ;SPINDLE SPEED
Q579=+1 ;PLUNGING FACTOR S
Q575=+0 ;INFEEED STRATEGY

- ▶ **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 direction of spindle rotation is taken into account:
+1 = climb milling
-1 = up-cut milling
PREDEF: The control uses the value from a **GLOBAL DEF** block. (if you enter 0, climb milling is performed)
- ▶ **Q576 Spindle speed?:** Spindle speed in revolutions per minute (rpm) for the roughing tool.
0: The speed from the **TOOL CALL** block is used
>0: If a value greater than zero is entered, then this speed is used Input range: 0 to 99999
- ▶ **Q579 Factor for plunging speed?** Factor by which the control reduces the **SPINDLE SPEED Q576** for downfeed into the material. Input range: 0.2 to 1.5
- ▶ **Q575 Infeed strategy (0/1)?:** Type of downfeed:
0: The contour is machined from top to bottom
1: The contour is machined from bottom to top. This strategy benefits from the maximum plunging depth.

10.4 OCM cutting data calculator (Option 167)

Fundamentals of the OCM cutting data calculator

Introduction

The OCM cutting data calculator is used to determine the Cutting data for Cycle **272 OCM ROUGHING**. These result from the properties of the material and the tool. The calculated cutting data help to achieve high material removal rates and therefore increase the productivity.

In addition, you can use the OCM cutting data calculator to specifically influence the load on the tool via sliders for the mechanical and thermal loads. This allows you to optimize the process reliability, the wear on the tool, and the productivity.

Prerequisites



Refer to your machine manual!

In order to capitalize on the calculated Cutting data, you need a sufficiently powerful spindle as well as a stable machine tool.

- The entered values are based on the assumption that the workpiece is firmly clamped in place.
- The entered values are based on the assumption that the tool is seated firmly in its holder.
- The tool being used must be appropriate for the material to be machined.



In case of large cutting depths and a large angle of twist, strong pulling forces develop in the direction of the tool axis. Make sure to have a sufficient finishing allowance for the floor.

Maintaining the cutting conditions

Use the cutting data only for Cycle **272 OCM ROUGHING**.

Only this cycle ensures that the permissible tool contact angle is not exceeded for the contours to be machined.

Chip removal

NOTICE

Caution: Danger to the tool and workpiece!

If the chips are not removed in an optimum manner, they could get caught in narrow pockets at these high metal removal rates. There is then a risk of tool breakage!

- Ensure that the chips are removed in an optimum manner, as recommended by the OCM cutting data calculator.

Process cooling

The OCM cutting data calculator recommends dry cutting with cooling by compressed air for most materials. The compressed air must be aimed directly at the cutting location. The best method is through the tool holder. If this is not possible, you can also mill with an internal coolant supply.

However, chip removal might not be as efficient when using tools with an internal coolant supply. This can lead to shortened tool life.

Operation

Opening the cutting data calculator

Open the cutting data calculator as follows:



- ▶ Edit Cycle **272 OCM ROUGHING**



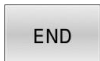
- ▶ Press the **OCM CUTTING DATA** soft key
- > The control opens the OCM cutting data calculator form.

Closing the cutting data calculator

Close the cutting data calculator as follows:



- ▶ Press **APPLY**
- > The control applies the determined Cutting data to the intended cycle parameters.
- > The current entries are stored, and are in place when the cutting data calculator is opened again.



- or
- ▶ Press the **END** or **CANCEL** soft key
 - > The current entries are not stored.
 - > The control does not apply any values to the cycle.



The OCM cutting data calculator calculates associated values for these cycle parameters:

- Plunging depth(Q202)
- Overlap factor(Q370)
- Spindle speed(Q576)
- Climb or up-cut(Q351)

When using the OCM cutting data calculator, you must not edit these parameters in the cycle later.

Fillable form

The control uses various colors in the fillable form:

- White background: entry required
- Red input values: missing or incorrect entry
- Gray background: no entry possible



The input fields of the workpiece material and the tool are gray. You can change them only through the selection list or the tool table.

Workpiece material

Proceed as follows to select the workpiece material:

- ▶ Tap the **Select** button
 - The control opens a selection list with various types of steel, aluminum, and titanium.
 - ▶ Select the workpiece material
- or
- ▶ Enter a search term in the search field
 - The control displays the materials or material groups that were found. Press the **RESET** button to switch back to the original selection list.
 - ▶ Apply your selection of the workpiece material with the **OK** button



If your material is not listed in the table, choose an appropriate material group or a material with similar cutting properties.

The selection list also shows the version number of your current workpiece-material table. You can update this if necessary. You will find the workpiece-material table **ocm.xml** in the **TNC:\system_calcprocess** directory.

Tool

You can choose the tool either by selecting it from the tool table **tool.t** or by entering the data manually.

Proceed as follows to select the tool:

- ▶ Tap the **Select** button
- > The control opens the active tool table **tool.t**.
- ▶ Select the tool
- ▶ Confirm with **OK**
- > The control applies the Diameter and the number of teeth entered in **tool.t**.
- ▶ Define the Angle of twist

Or proceed as follows without selecting a tool:

- ▶ Enter the Diameter
- ▶ Define the number of teeth
- ▶ Enter the Angle of twist

T	NAME	R	DR	CUT
0	MILLWERKZEUG	+0	+0	0
1	MILL_D2_ROUGH	+1	+0	2
2	MILL_D4_ROUGH	+2	+0	2
3	MILL_D6_ROUGH	+3	+0	3
4	MILL_D8_ROUGH	+4	+0	3
5	MILL_D10_ROUGH	+5	+0	3
6	MILL_D12_ROUGH	+6	+0	4
7	MILL_D14_ROUGH	+7	+0	4
8	MILL_D16_ROUGH	+8	+0	4
9	MILL_D18_ROUGH	+9	+0	4
10	MILL_D20_ROUGH	+10	+0	4
11	MILL_D22_ROUGH	+11	+0	4
12	MILL_D24_ROUGH	+12	+0	4
13	MILL_D26_ROUGH	+13	+0	4
14	MILL_D28_ROUGH	+14	+0	4
15	MILL_D30_ROUGH	+15	+0	4
16	MILL_D32_ROUGH	+16	+0	4
17	MILL_D34_ROUGH	+17	+0	4
18	MILL_D36_ROUGH	+18	+0	4
19	MILL_D38_ROUGH	+19	+0	4

Input dialog	Description
Diameter	Diameter of the roughing tool in mm (input range: 1 mm to 40 mm) Value is applied automatically after the roughing tool has been selected.
Number of teeth	Number of teeth of the roughing tool (input range: 1 to 10) Value is applied automatically after the roughing tool has been selected.
Angle of twist	Angle of twist of the roughing tool in ° (input range: 0° to 80°) If there are different angles of twist, then enter the average value.



You can modify the values of the Diameter and the number of teeth at any time. The modified value is **not** written to the tool table **tool.t**!

You will find the Angle of twist in the description of your tool, for example in the tool catalog of the tool manufacturer.


Limits

For the Limits, you need to define the maximum spindle speed and the maximum milling speed. The calculated Cutting data are then limited to these values.

Input dialog	Description
Max. spindle speed	Maximum spindle speed in rpm permitted by the machine and the clamping situation:
Max. milling speed	Maximum milling speed (feed rate) in mm/min permitted by the machine and the clamping situation:

Process parameters

For the Process parameters, you need to define the Plunging depth(Q202) as well as the mechanical and thermal loads:

Input dialog	Description
Plunging depth(Q202)	Plunging depth (>0 mm to [6 times the tool diameter]) The value from cycle parameter Q202 is applied when starting the OCM cutting data calculator.
Mechanical load on tool	Slider for selection of the mechanical load (the value is normally between 70 % and 100 %)
Thermal load on tool	<p>Slider for selection of the thermal load Set the slider according to the thermal wear-resistance (coating) of your tool.</p> <ul style="list-style-type: none"> ■ HSS: low thermal wear-resistance ■ VHM (uncoated or normally-coated solid carbide milling cutters): medium thermal wear-resistance ■ Coated (fully-coated solid carbide milling cutters): high thermal wear-resistance <div style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p> The slider is only effective in the range highlighted in green. This limiting depends on the maximum spindle speed, the maximum feed rate, and the selected material. If the slider is in the red range, the control will use the maximum permissible value.</p> </div>

Further information: "Process parameters ", Page 301

Cutting data

The control displays the calculated values in the Cutting data section.

The following Cutting data are applied to the appropriate cycle parameters in addition to the plunging depth **Q202**:

Cutting data:	Applied to cycle parameter:
Overlap factor(Q370)	Q370 = TOOL PATH OVERLAP
Milling feed(Q207) in mm/min	Q207 = FEED RATE MILLING
Spindle speed(Q576) in rpm	Q576 = SPINDLE SPEED
Climb or up-cut(Q351)	Q351= CLIMB OR UP-CUT



The OCM cutting data calculator calculates values only for climb milling (**Q351=+1**). For this reason, it always applies **Q351=+1** to the cycle parameter.

The following cutting data is for informational purposes and recommendation:

- Lateral infeed in mm
- Tooth feed FZ in mm
- Cutting speed VC in m/min
- Material removal rate in cm³/min
- Spindle power in kW
- Recommended cooling

These values help you assess whether your machine tool is able to meet the selected cutting conditions.

Process parameters

The two sliders for the mechanical and thermal load have an influence on the process forces and temperatures prevalent on the cutting edges. Higher values increase the metal removal rate, but also lead to a higher load. Moving the sliders makes different process parameters possible.

Maximum material removal rate

For a maximum material removal rate, set the slider for the mechanical load to 100 % and the slider for the thermal load according to the coating of your tool.

If the defined limitations permit it, the cutting data utilize the tool at its mechanical and thermal load capacities. For large tool diameters ($D \geq 16$ mm), a very high level of spindle power can be necessary.

For the theoretically expectable spindle power, refer to the cutting data output.



If the permissible spindle power is exceeded, you can first move the slider for the mechanical load to a lower value. If necessary, you can also reduce the plunging depth (a_p).

Please note that at very high shaft speeds, a spindle running below its rated speed will not attain the rated power.

If you wish to achieve a high material removal rate, you must ensure that chips are removed optimally.

Reduced load and low wear

In order to decrease the mechanical load and the thermal wear, reduce the mechanical load to 70 %. Reduce the thermal load to a value that corresponds to 70 % of the coating of your tool.

These settings utilize the tool in a manner that is mechanically and thermally balanced. In general the tool will then reach its maximum service life. The lower mechanical load makes a smoother process possible that is less subject to vibration.

Achieving an optimum result

If the Cutting data do not lead to a satisfactory cutting process, then different causes might be the reason for this.

Excessively high mechanical load

If there is an excessive mechanical load, you must first reduce the process force.

The following conditions are indications of excessive mechanical load:

- Cutting edges of the tool break
- Shaft of the tool breaks
- Excessive spindle torque or spindle power
- Excessive axial or radial forces on the spindle bearing
- Undesired oscillations or chatter
- Oscillations due to weak clamping
- Oscillations due to long projecting tool

Excessively high thermal load

If there is an excessive thermal load, you must reduce the process temperature.

The following conditions indicate an excessive thermal load on the tool:

- Excessive crater wear at the cutting surface
- The tool glows
- The cutting edges melt (for very materials that are very difficult to cut, such as titanium)

Material removal rate is too low

If the machining time is too long and it must be reduced, the material removal rate can be increased by moving both sliders.

If both the machine and the tool still have potential, then it is recommended that the slider for the process temperature be raised to a higher value first. Subsequently, if possible, you can also raise the slider for the process forces to a higher value.

Remedies for problems

The table below provides an overview of possible types of problems as well as countermeasures for them.

Condition	Slider Mechanical load on tool	Slider Thermal load on tool	Miscellaneous
Vibrations (such as weak clamping or tools that project too far)	Decrease	Perhaps increase	Check the clamping
Undesired vibrations or chatter	Decrease	-	
Shaft of tool breaks	Decrease	-	Check the chip removal
Cutting edges of the tool break	Decrease	-	Check the chip removal
Excessive wear	Perhaps increase	Decrease	
The tool glows	Perhaps increase	Decrease	Check the cooling
Machining time is too long	Perhaps increase	Increase this first	
Excessive spindle load	Decrease	-	
Excessive axial force on spindle bearing	Decrease	-	<ul style="list-style-type: none"> ■ Reduce the plunging depth ■ Use a tool with a lower angle of twist
Excessive radial force on spindle bearing	Decrease	-	

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

Application

With Cycle **273 OCM FINISHING FLOOR**, you can program finishing with the finishing allowance for the floor programmed in Cycle **271**.

Requirements

Before programming the call of Cycle **273**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable

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 **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 mills off the material remaining from rough-out (finishing allowance)
- 5 Finally, the tool retracts in the tool axis to the clearance height

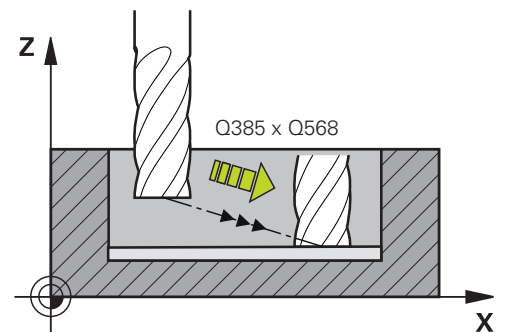
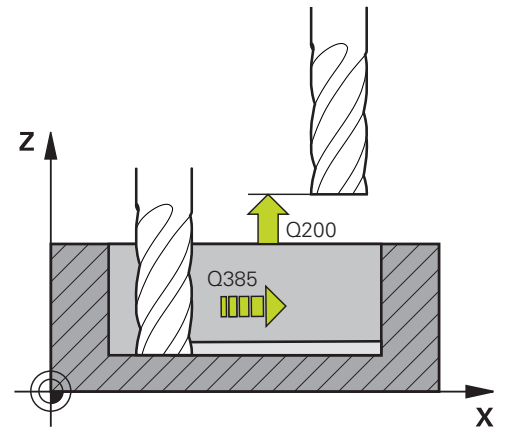
Please note while programming!

- 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.
- If you do not define parameter **Q438 ROUGH-OUT TOOL**, the control will display an error message.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- If you use an overlap factor greater than 1, residual material may be left over. Check the contour using the program verification graphics and slightly change the overlap factor, if necessary. This allows another distribution of cuts, which often provides the desired results.

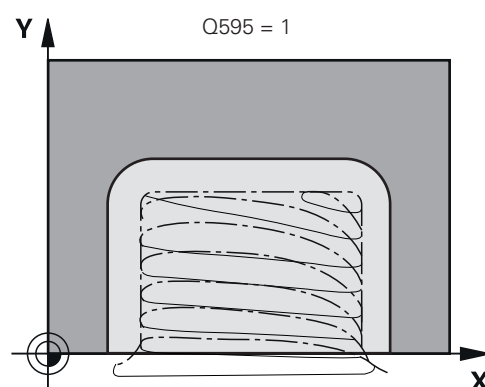
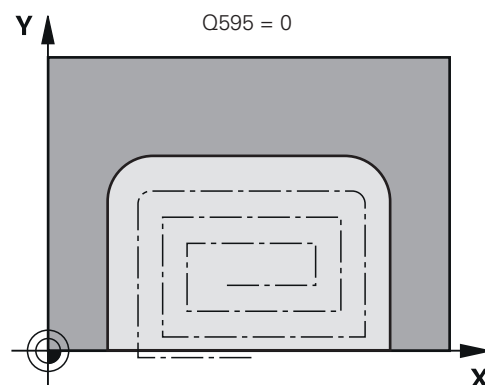
Cycle parameters



- ▶ **Q370 Path overlap factor?:** $Q370 \times \text{tool radius} = \text{stepover factor } k$. 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 in mm/min. This feed rate will be used below the coordinate surface, but outside the defined material.
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. You can 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.
Q438=-1: The control assumes that the tool last used was the rough-out tool (default behavior)
Input range if a number is entered: -1 to +32767.9



- **Q595 Strategy (0/1)?**: Machining strategy for finishing
0: Equidistant = strategy with constant distances between paths
1: Strategy with constant tool angle
- **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



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
Q595=+1 ;STRATEGY
Q577=+0.2 ;APPROACH RADIUS FACTOR

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

Application

With Cycle **274 OCM FINISHING SIDE**, you can program finishing with the side finishing allowance programmed in Cycle **271**. You can run this cycle in climb or up-cut milling.

Requirements

Before programming the call of Cycle **274**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable

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 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

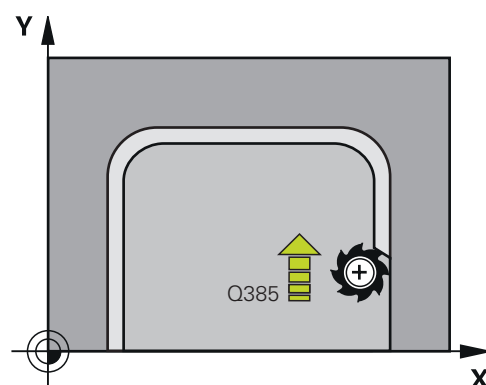
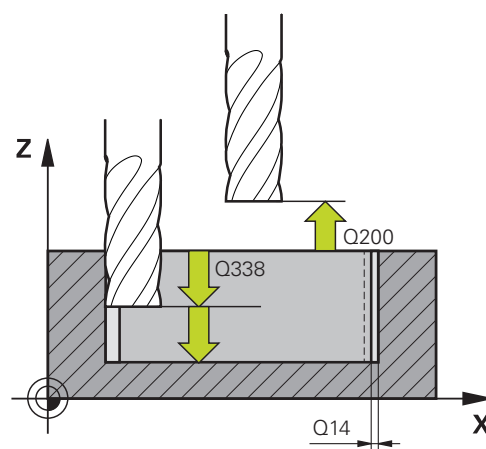
Please note while programming!

- 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**.
- If you do not define parameter **Q438 ROUGH-OUT TOOL**, the control will display an error message.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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 in mm/min. This feed rate will be used below the coordinate surface, but outside the defined material.
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. You can 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.
Q438=-1: The control assumes that the tool last used was the rough-out tool (default behavior)
Input range if a number is entered: -1 to +32767.9
- ▶ **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
PREDEF: The control uses the value from a **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

10.7 OCM CHAMFERING (Cycle 277, DIN/ISO: G277, Option 167)

Application

Cycle **277 OCM CHAMFERING** enables you to deburr edges of complex contours that you roughed out using OCM cycles.

This cycle considers adjacent contours and boundaries that you called before with Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements.

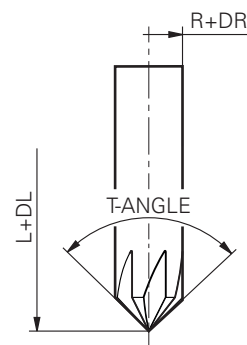
Requirements

Before the control can execute Cycle **277**, you need to create the tool in the tool table using appropriate parameters:

- **L + DL**: Overall length up to the theoretical tip
- **R + DR**: Definition of the overall tool radius
- **T-ANGLE**: Point angle of the tool

In addition, you need to program other cycles before programming the call of Cycle **277**:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable



Cycle run

- 1 The tool uses rapid traverse to move to **Q260 CLEARANCE HEIGHT**. The control takes this information from Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements
- 2 The tool then moves to the starting point. This point is determined automatically based on the programmed contour.
- 3 In the next step, the tool moves at **FMAX** to set-up clearance **Q200**
- 4 Then, the tool plunges vertically to **Q353 DEPTH OF TOOL TIP**
- 5 The tool approaches the contour in a tangential or vertical movement (depending on the available space). For machining the chamfer, the tool uses the milling feed rate **Q207**
- 6 Then, the tool is retracted from the contour in a tangential or vertical movement (depending on the available space).
- 7 If there are several contours, the control positions the tool at clearance height after each contour and then moves it to the next starting point. Steps 3 to 6 are repeated until the programmed contour is completely chamfered
- 8 At the end of machining, the tool is retracted along the tool axis and moves to **Q260 CLEARANCE HEIGHT**

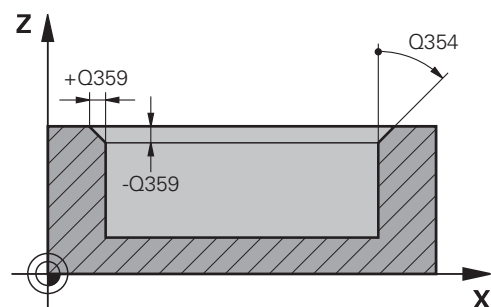
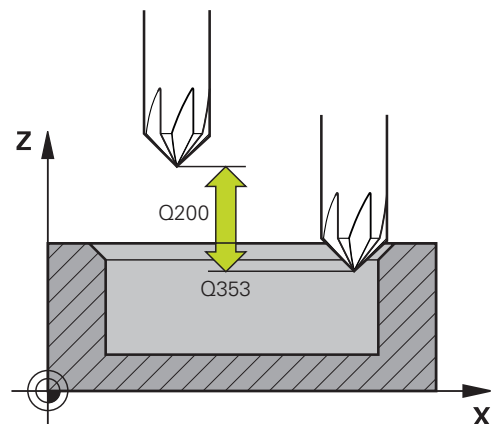
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for chamfering. The starting point depends on the available space.
- If the value of parameter **Q353 DEPTH OF TOOL TIP** is less than the value of parameter **Q359 CHAMFER WIDTH**, the control will display an error message.
- If you do not define parameter **Q438 ROUGH-OUT TOOL**, the control will display an error message.
- Measure the length of the tool up to the theoretical tool tip.
- The control monitors the tool radius. Adjacent walls machined with Cycle **271 OCM CONTOUR DATA** or with the **12xx** shape cycles will remain intact.
- Remember that the control does not monitor the theoretical tool tip for collision. In the **Test Run** operating mode, the control always uses the theoretical tool tip for the simulation. This can result, e.g. for tools without an actual tool tip, in simulated contour damages while the NC program is actually error-free.

Cycle parameters



- ▶ **Q353 Depth of tool tip?** (incremental): Distance between theoretical tool tip and workpiece surface coordinates.
Input range: -999.9999 to -0.0001
- ▶ **Q359 Width of chamfer (-/+)?** (incremental):
Width or depth of the chamfer:
-: Depth of the chamfer
+: Width of the chamfer
Input range: -999.9999 to +999.9999
- ▶ **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**
- ▶ **Q253 Feed rate for pre-positioning?** Traversing speed of the tool during positioning in mm/min.
Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface.
Input range: 0 to 99999.9999; alternatively **PREDEF**
- ▶ **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. You can 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.
Q438=-1: The control assumes that the tool last used was the rough-out tool (default behavior)
Input range if a number is entered: -1 to +32767.9
- ▶ **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
PREDEF: The control uses the value from a **GLOBAL DEF** block. (if you enter 0, climb milling is performed)
- ▶ **Q354 Angle of chamfer?** Chamfer angle
0: The chamfer angle is half the defined **T-ANGLE** from the tool table
>0: The chamfer angle is compared to the value of **T-ANGLE** from the tool table. If these two values do not match, the control will display an error message.
Input range: 0 to 89



Example

59 CYCL DEF 277 OCM CHAMFERING	
Q353=-1	;DEPTH OF TOOL TIP
Q359=+0.2	;CHAMFER WIDTH
Q207=+500	;FEED RATE MILLING
Q253=+750	;F PRE-POSITIONING
Q200=+2	;SET-UP CLEARANCE
Q438=-1	;ROUGH-OUT TOOL
Q351=+1	;CLIMB OR UP-CUT
Q354=+0	;CHAMFER ANGLE

10.8 OCM standard shapes

Fundamentals

The control provides cycles for frequently used shapes. You can program these shapes as pockets, islands, or boundaries.

These shape cycles offer the following benefits:

- You can conveniently program the shapes and machining data without the need to program an individual path contour.
- Frequently needed shapes can be reused.
- If you want to program an island or an open pocket, the control provides you with more cycles for defining the shape boundary.
- The Boundary shape type enables you to face-mill your shape.

With a shape, you can redefine the OCM contour data and cancel the definition of a previously defined Cycle **271 OCM CONTOUR DATA** or of a shape boundary.

For defining shapes, the control provides the following cycles:

- **1271 OCM RECTANGLE**, see Page 313
- **1272 OCM CIRCLE**, see Page 316
- **1273 OCM SLOT / RIDGE**, see Page 318
- **1278 OCM POLYGON**, see Page 320

For defining shape boundaries, the control provides the following cycles:

- **1281 OCM RECTANGLE BOUNDARY**, see Page 323
- **1282 OCM CIRCLE BOUNDARY**, see Page 325

10.9 OCM RECTANGLE (Cycle 1271, DIN/ISO: G1271, Option 167)

Application

Use shape cycles **1271 OCM RECTANGLE** to program a rectangle. You can use the shape to machine a pocket, an island, or a boundary by face milling.

If you work with Cycle **1271**, program the following:

- Cycle **1271 OCM RECTANGLE**
 - If you program **Q650=1** (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

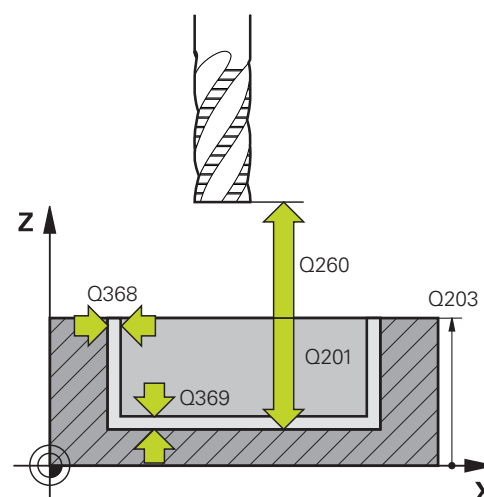
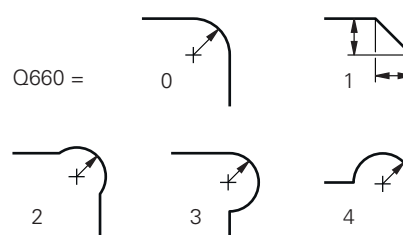
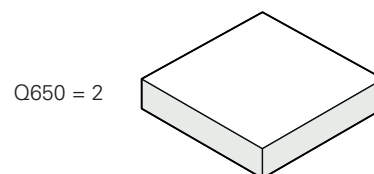
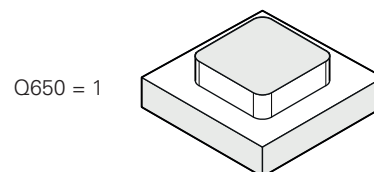
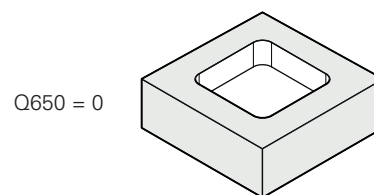
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1271** 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 **1271** are valid for the OCM machining cycles **272** to **274** and **277**.
- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters



- ▶ **Q650 Type of figure?:** Geometry of the shape.
 0: Pocket
 1: Island
 2: Boundary for face milling
- ▶ **Q218 First side length?** (incremental): Length of the first side of the shape, parallel to the principal axis.
 Input range: 0 to 99999.9999
- ▶ **Q219 Second side length?** (incremental): Length of the 2nd side of the shape, parallel to the secondary axis.
 Input range: 0 to 99999.9999
- ▶ **Q660 Type of corners?:** Geometry of the corners:
 0: Radius
 1: Chamfer
 2: Milling corners in the principal and secondary axis directions
 3: Milling corners in the principal axis direction
 4: Milling corners in the secondary axis direction
- ▶ **Q220 Corner radius?:** Radius or chamfer of the corner of the shape.
 Input range: 0 to 99999.9999
- ▶ **Q367 Position of pocket (0/1/2/3/4)?:** Position of the pocket with respect to the tool position when the cycle is called:
 0: Tool position = center of pocket
 1: Tool position = lower left corner
 2: Tool position = lower right corner
 3: Tool position = upper right corner
 4: Tool position = upper left corner
- ▶ **Q224 Angle of rotation?** (absolute): Angle by which the shape will be rotated. The center of rotation is at the center of the shape.
 Input range: -360 to +360
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface with reference to the active preset
 Input range: -99999.9999 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between the workpiece surface and the 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): Coordinate in the tool axis 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

Example

59 CYCL DEF 1271 OCM RECTANGLE	
Q650=+1	;FIGURE TYPE
Q218=+60	;FIRST SIDE LENGTH
Q219=+40	;2ND SIDE LENGTH
Q660=+0	;CORNER TYPE
Q220=+0	;CORNER RADIUS
Q367=+0	;POCKET POSITION
Q224=+0	;ANGLE OF ROTATION
Q203=+0	;SURFACE COORDINATE
Q201=-10	;DEPTH
Q368=+0	;ALLOWANCE FOR SIDE
Q369=+0	;ALLOWANCE FOR FLOOR
Q260=+50	;CLEARANCE HEIGHT
Q578=+0.2	;INSIDE CORNER FACTOR

10.10 OCM CIRCLE (Cycle 1272, DIN/ISO: G1272, Option 167)

Application

Use shape cycle **1272 OCM CIRCLE** to program a circle. You can use the shape to machine a pocket, an island, or a boundary by face milling.

If you work with Cycle **1272**, program the following:

- Cycle **1272 OCM CIRCLE**
 - If you program **Q650=1** (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

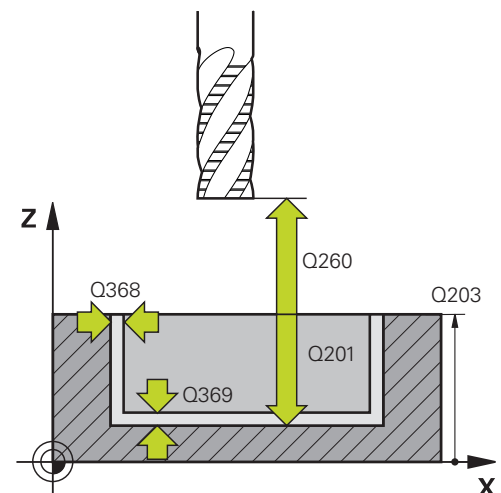
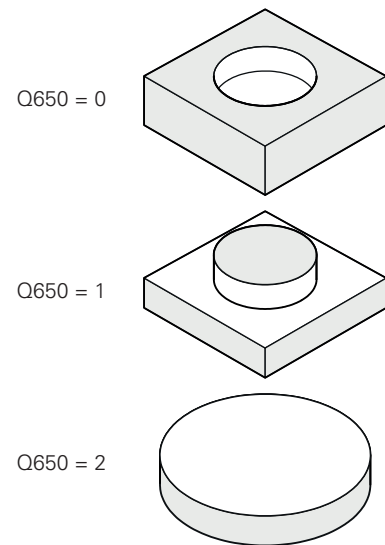
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1272** 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 **1272** are valid for the OCM machining cycles **272** to **274** and **277**.
- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters



- ▶ **Q650 Type of figure?:** Geometry of the shape.
 0: Pocket
 1: Island
 2: Boundary for face milling
- ▶ **Q223 Circle diameter?:** Diameter of the finished circle.
 Input range: 0 to 99999.9999
- ▶ **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 = center of shape
 1: Tool position = quadrant transition at 90°
 2: Tool position = quadrant transition at 0°
 3: Tool position = quadrant transition at 270°
 4: Tool position = quadrant transition at 180°
- ▶ **Q203 Workpiece surface coordinate?** (absolute):
 Coordinate of the workpiece surface with reference to the active preset
 Input range: -99999.9999 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between the workpiece surface and the 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): Coordinate in the tool axis 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 minimum radius of a circular pocket results from the tool radius plus the product of tool radius and **Q578**.
 Input range: 0.05 to 0.99



Example

59 CYCL DEF 1272 OCM CIRCLE	
Q650=+0	;FIGURE TYPE
Q223=+50	;CIRCLE DIAMETER
Q367=+0	;POCKET POSITION
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

10.11 OCM SLOT / RIDGE (Cycle 1273, DIN/ISO: G1273, Option 167)

Application

Use shape cycle **1273 OCM SLOT / RIDGE** to program a slot or a ridge. This shape cycle also enables you to program a boundary for face milling.

If you work with Cycle **1273**, program the following:

- Cycle **1273 OCM SLOT / RIDGE**
 - If you program **Q650=1** (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

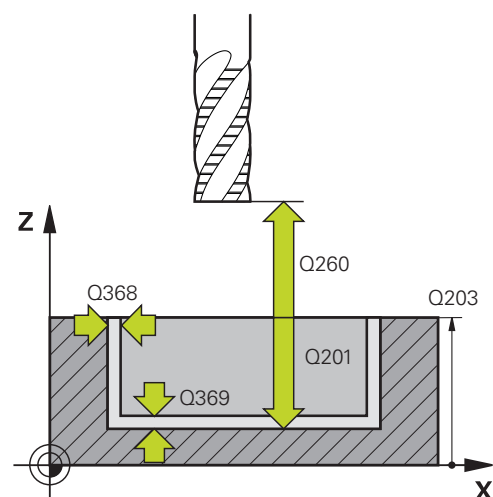
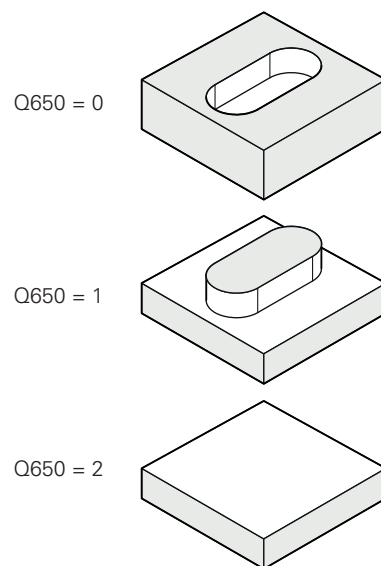
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1273** 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 **1273** are valid for the OCM machining cycles **272** to **274** and **277**.
- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters



- ▶ **Q650 Type of figure?:** Geometry of the shape.
 0: Pocket
 1: Island
 2: Boundary for face milling
- ▶ **Q219 Width of slot?** (incremental): Width of the slot or ridge, parallel to the secondary axis of the working plane.
 Input range: 0 to 99999.9999
- ▶ **Q218 Length of slot?** (incremental): Length of the slot or ridge, parallel to the principal axis of the working plane.
 Input range: 0 to 99999.9999
- ▶ **Q367 Position of slot (0/1/2/3/4)?:** Position of the slot with respect to the tool position when the cycle is called:
 0: Tool position = center of slot
 1: Tool position = left end of slot
 2: Tool position = center of left slot arc
 3: Tool position = center of right slot arc
 4: Tool position = right end of slot
- ▶ **Q224 Angle of rotation?** (absolute): Angle by which the shape will be rotated. The center of rotation is at the center of the shape.
 Input range: -360 to +360
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface with reference to the active preset
 Input range: -99999.9999 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between the workpiece surface and the 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): Coordinate in the tool axis 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 minimum radius (slot width) of a slot results from the tool radius plus the product of tool radius and **Q578**.
 Input range: 0.05 to 0.99



Example

59 CYCL DEF 1273 OCM SLOT / RIDGE
Q650=+0 ;FIGURE TYPE
Q219=+10 ;SLOT WIDTH
Q218=+60 ;SLOT LENGTH
Q367=+0 ;SLOT POSITION
Q224=+0 ;ANGLE OF ROTATION
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

10.12 OCM POLYGON (Cycle 1278, DIN/ISO: G1278, Option 167)

Application

Use shape cycle **1278 OCM POLYGON** to program a polygon. You can use the shape to machine a pocket, an island, or a boundary by face milling.

If you work with Cycle **1278**, program the following:

- Cycle **1278 OCM POLYGON**
 - If you program **Q650=1** (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

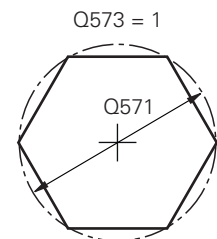
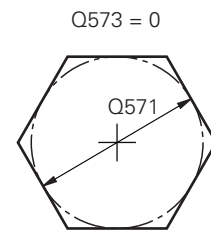
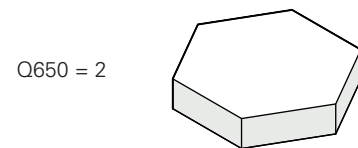
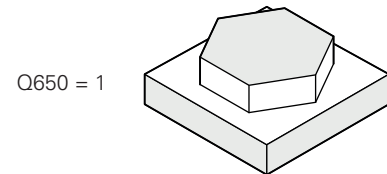
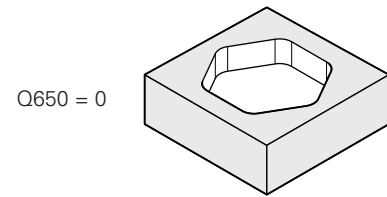
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1278** 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 **1278** are valid for the OCM machining cycles **272** to **274** and **277**.
- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

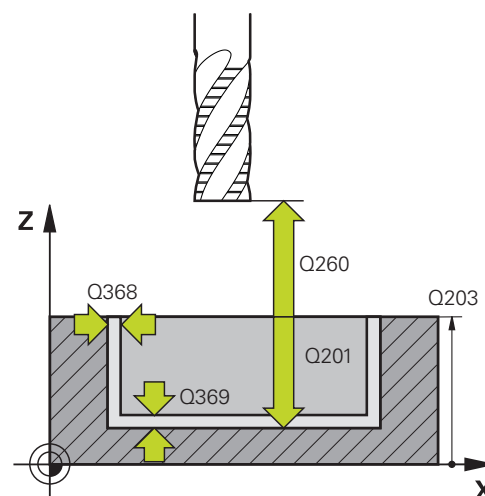
Cycle parameters



- ▶ **Q650 Type of figure?:** Geometry of the shape.
 0: Pocket
 1: Island
 2: Boundary for face milling
- ▶ **Q573 Inscr.circle/circumcircle (0/1)?:** Definition of whether the dimension **Q571** shall reference the inscribed circle or the perimeter:
 0= dimension 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 entered here references the inscribed circle or the perimeter.
 Input range: 0 to 99999.9999
- ▶ **Q572 Number of corners?:** Enter the number of corners of the polygon. The control will always distribute the corners evenly on the polygon.
 Input range: 3 to 30
- ▶ **Q660 Type of corners?:** Geometry of the corners:
 0: Radius
 1: Chamfer
- ▶ **Q220 Corner radius?:** Radius or chamfer of the corner of the shape.
 Input range: 0 to 99999.9999
- ▶ **Q224 Angle of rotation?** (absolute): Angle by which the shape will be rotated. The center of rotation is at the center of the shape.
 Input range: -360 to +360
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface with reference to the active preset
 Input range: -99999.9999 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between the workpiece surface and the 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): Coordinate in the tool axis 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



Example

59 CYCL DEF 1278 OCM POLYGON	
Q650=+0	;FIGURE TYPE
Q573=+0	;REFERENCE CIRCLE
Q571=+50	;REF-CIRCLE DIAMETER
Q572=+6	;NUMBER OF CORNERS
Q660=+0	;CORNER TYPE
Q220=+0	;CORNER RADIUS
Q224=+0	;ANGLE OF ROTATION
Q203=+0	;SURFACE COORDINATE
Q201=-10	;DEPTH
Q368=+0	;ALLOWANCE FOR SIDE
Q369=+0	;ALLOWANCE FOR FLOOR
Q260=+50	;CLEARANCE HEIGHT
Q578=+0.2	;INSIDE CORNER FACTOR

10.13 OCM RECTANGLE BOUNDARY (Cycle 1281, DIN/ISO: G1281, Option 167)

Application

Use Cycle **1281 OCM RECTANGLE BOUNDARY** to program a rectangular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard shape.

The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE** = 0 (pocket) or = 1 (island) within an OCM standard shape cycle.

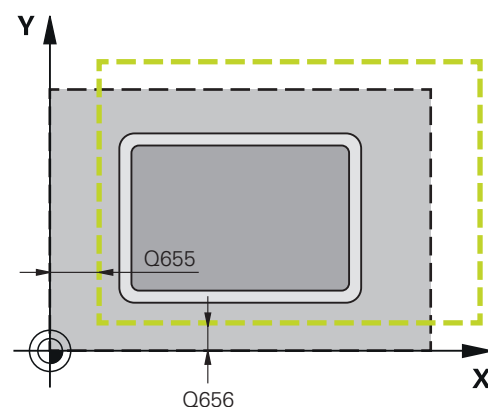
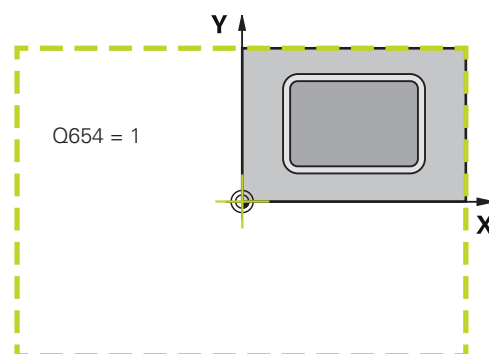
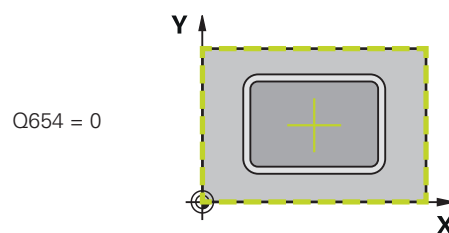
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1281** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1281** are valid for Cycles **1271** to **1273** and **1278**.

Cycle parameters



- ▶ **Q651 Length of major axis?:** Length of the first boundary side, parallel to the principal axis.
Input range: 0.001 to 9999.999
- ▶ **Q652 Length of minor axis?:** Length of the 2nd boundary side, parallel to the secondary axis.
Input range: 0.001 to 9999.999
- ▶ **Q654 Position reference for figure?:** Specify the center position reference:
0: The center of the boundary references the center of the machining contour
1: The center of the boundary references the datum
- ▶ **Q655 Shift in major axis?:** Shift of the rectangle boundary along the principal axis.
Input range: -999.999 to +999.999
- ▶ **Q656 Shift in minor axis?:** Shift of the rectangle boundary along the secondary axis.
Input range: -999.999 to +999.999



Example

59 CYCL DEF 1281 OCM RECTANGLE BOUNDARY
Q651=+50 ;LENGTH 1
Q652=+50 ;LENGTH 2
Q654=+0 ;POSITION REFERENCE
Q655=+0 ;SHIFT 1
Q656=+0 ;SHIFT 2

10.14 OCM CIRCLE BOUNDARY (Cycle 1282, DIN/ISO: G1282, Option 167)

Application

Use Cycle **1282 OCM CIRCLE BOUNDARY** to program a circular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard shape.

The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE = 0** (pocket) or **= 1** (island) in an OCM standard shape cycle.

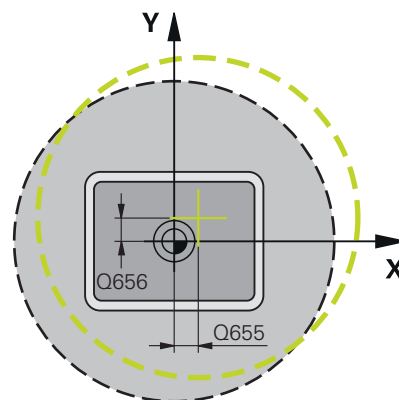
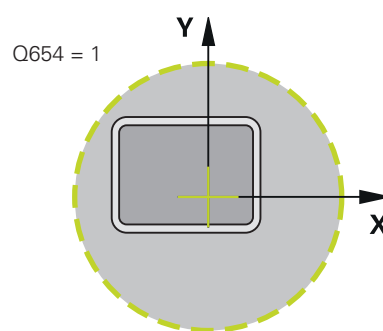
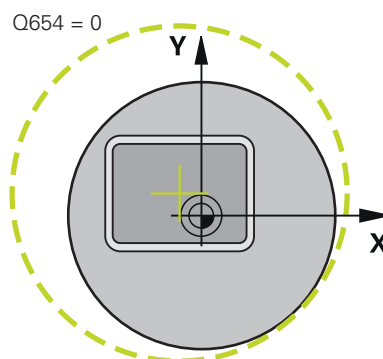
Please note while programming!

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1282** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1282** are valid for Cycles **1271** to **1273** and **1278**.

Cycle parameters



- ▶ **Q653 Diameter?:** Diameter of the circular bounding frame.
Input range: 0.001 to 9999.999
- ▶ **Q654 Position reference for figure?:** Specify the center position reference:
0: The center of the boundary references the center of the machining contour
1: The center of the boundary references the datum
- ▶ **Q655 Shift in major axis?:** Shift of the rectangle boundary along the principal axis.
Input range: -999.999 to +999.999
- ▶ **Q656 Shift in minor axis?:** Shift of the rectangle boundary along the secondary axis.
Input range: -999.999 to +999.999



Example

```
59 CYCL DEF 1282 OCM CIRCLE
BOUNDARY
```

```
Q653=+50 ;DIAMETER
```

```
Q654=+0 ;POSITION REFERENCE
```

```
Q655=+0 ;SHIFT 1
```

```
Q656=+0 ;SHIFT 2
```

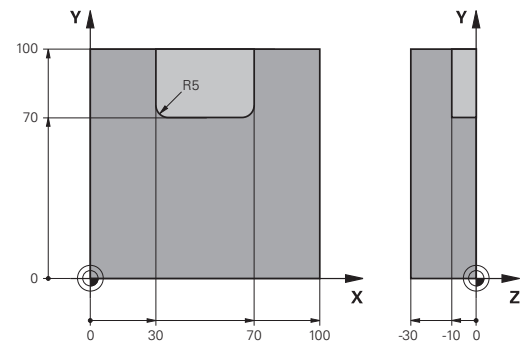
10.15 Programming examples

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 that is defined by means of an island and a boundary. Machining includes roughing and finishing of an open pocket.

Program sequence

- Tool call: roughing cutter Ø 20 mm
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: roughing cutter Ø 8 mm
- Define and call Cycle **272**
- Tool call: finishing cutter Ø 6 mm
- 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 mm
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=+10 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207=+6500 ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253= AUTO ;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	

Q576=+6500	;SPINDLE SPEED	
Q579=+0.7	;PLUNGING FACTOR S	
Q575=+0	;INFEEED STRATEGY	
10 CYCL CALL		Cycle call
11 TOOL CALL "MILL_D8" Z S8000 F1500		Tool call, diameter 8 mm
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=+10	;PLUNGING DEPTH	
Q370=+0.4	;TOOL PATH OVERLAP	
Q207=+6000	;FEED RATE MILLING	
Q568=+0.6	;PLUNGING FACTOR	
Q253= AUTO	;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	
Q576=+10000	;SPINDLE SPEED	
Q579=+0.7	;PLUNGING FACTOR S	
Q575=+0	;INFEEED STRATEGY	
16 CYCL CALL		Cycle call
17 TOOL CALL "MILL_D6_FINISH" Z S10000 F2000		Tool call, diameter 6 mm
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	;INFEEED 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

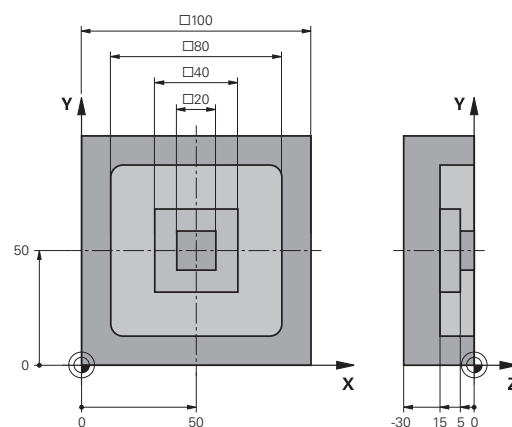
25 M30	End of program
26 LBL 1	Contour subprogram 1
27 L X+0 Y+0	
28 L X+100	
29 L Y+100	
30 L X+0	
31 L Y+0	
32 LBL 0	
33 LBL 2	Contour subprogram 2
34 L X+0 Y+0	
35 L X+100	
36 L Y+100	
37 L X+70	
38 L Y+70	
39 RND R5	
40 L X+30	
41 RND R5	
42 L Y+100	
43 L X+0	
44 L Y+0	
45 LBL 0	
46 END PGM OCM_POCKET MM	

Example: Program various depths with OCM cycles

The following NC program illustrates the use of OCM cycles. You will define one pocket and two islands at different heights. Machining includes roughing and finishing of a contour.

Program sequence

- Tool call: roughing cutter Ø 10 mm
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: finishing cutter Ø 6 mm
- 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 10 mm
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=+20 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207=+6500 ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253= AUTO ;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	
Q576=+10000 ;SPINDLE SPEED	
Q579=+0.7 ;PLUNGING FACTOR S	
Q575=+1 ;INFEEED STRATEGY	
9 CYCL CALL	Cycle call
10 TOOL CALL "MILL_D6_FINISH" Z S10000 F2000	Tool call, diameter 6 mm

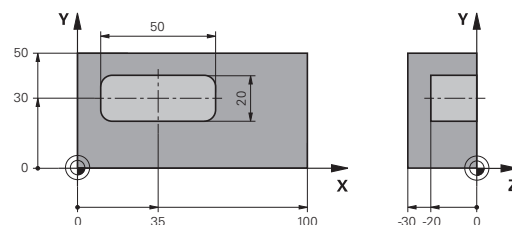
11 M3	
12 L Z+250 R0 FMAX	
13 L X+0 Y+0 R0 FMAX	
14 CYCL DEF 273 OCM FINISHING FLOOR	Define the floor finishing cycle
Q370=+0.8 ;TOOL PATH OVERLAP	
Q385= AUTO ;FINISHING FEED RATE	
Q568=+0.3 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=-1 ;ROUGH-OUT TOOL	
15 CYCL CALL	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	

Example: Face milling and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will face-mill a surface which will be defined by means of a boundary and an island. In addition, you will mill a pocket that contains an allowance for a smaller roughing tool.

Program sequence

- Tool call: roughing cutter Ø 12 mm
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: roughing cutter Ø 8 mm
- Define Cycle **272** and call it again



0 BEGIN PGM FACE_MILL MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+50 Z+2	
3 TOOL CALL "MILL_D12" Z S5000 F3000	Tool call, diameter 12 mm
4 CONTOUR DEF	
P1 = LBL "FRAME" I2 = LBL "FRAME" DEPTH2	
P3 = LBL "POCKET";	
5 CYCL DEF 271 OCM CONTOUR DATA	Define machining parameters
Q203=+2 ;SURFACE COORDINATE	
Q201=-22 ;DEPTH	
Q368=+0 ;ALLOWANCE FOR SIDE	
Q369=+0 ;ALLOWANCE FOR FLOOR	
Q260=+100 ;CLEARANCE HEIGHT	
Q578=+0.2 ;INSIDE CORNER FACTOR	
Q569=+1 ;OPEN BOUNDARY	
6 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+24 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207=+8000 ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253= AUTO ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=-1 ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
Q576=+8000 ;SPINDLE SPEED	
Q579=+0.7 ;PLUNGING FACTOR S	
Q575=+0 ;INFED STRATEGY	
7 L Z+100 R0 FMAX M3	
8 L X+0 Y+0 R0 FMAX M99	Cycle call
9 TOOL CALL "MILL_D8" Z S6000 F4000	Tool call, diameter 8 mm

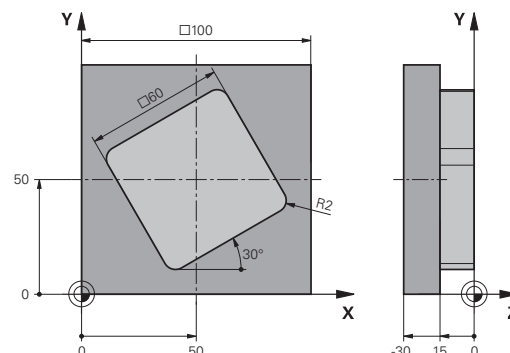
10 CYCL DEF 272 OCM ROUGHING	Define fine roughing using a roughing cycle
Q202=+25 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= 6500 ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253= AUTO ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
QS438="MILL_D12";ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
Q576=+10000 ;SPINDLE SPEED	
Q579=+0.7 ;PLUNGING FACTOR S	
Q575=+0 ;INFEED STRATEGY	
11 L Z+100 R0 FMAX M3	
12 L X+0 Y+0 R0 FMAX M99	Cycle call
13 M30	End of program
14 LBL "FRAME"	Contour subprogram FRAME
15 L X+0 Y+0	
16 L Y+50	
17 L X+100	
18 L Y+0	
19 L X+0	
20 LBL 0	
21 LBL "POCKET"	Contour subprogram POCKET
22 L X+10 Y+30	
23 L Y+40	
24 RND R5	
25 L X+60	
26 RND R5	
27 L Y+20	
28 RND R5	
29 L X+10	
30 RND R5	
31 L Y+30	
32 LBL 0	
33 END PGM FACE_MILL MM	

Example: Contour with OCM shape cycles

The following NC program illustrates the use of OCM cycles. Machining includes roughing and finishing of a island.

Program sequence

- Tool call: roughing cutter Ø 8 mm
- Define Cycle **1271**
- Define Cycle **1281**
- Define and call Cycle **272**
- Tool call: finishing cutter Ø 8 mm
- Define and call Cycle **273**
- Define and call Cycle **274**



0 BEGIN PGM OCM_FIGURE 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_D8" Z S8000 F1500	Tool call, diameter 8 mm
4 L Z+250 R0 FMAX M3	
5 CYCL DEF 1271 OCM RECTANGLE	Define OCM shape
Q650=+1 ;FIGURE TYPE	
Q218=+60 ;FIRST SIDE LENGTH	
Q219=+60 ;2ND SIDE LENGTH	
Q660=+0 ;CORNER TYPE	
Q220=+2 ;CORNER RADIUS	
Q367=+0 ;POCKET POSITION	
Q224=+30 ;ANGLE OF ROTATION	
Q203=+0 ;SURFACE COORDINATE	
Q201=-10 ;DEPTH	
Q368=+0.5 ;ALLOWANCE FOR SIDE	
Q369=+0.5 ;ALLOWANCE FOR FLOOR	
Q260=+100 ;CLEARANCE HEIGHT	
Q578=+0.2 ;INSIDE CORNER FACTOR	
6 CYCL DEF 1281 OCM RECTANGLE BOUNDARY	Define rectangle boundary
Q651=+100 ;LENGTH 1	
Q652=+100 ;LENGTH 2	
Q654=+0 ;POSITION REFERENCE	
Q655=+0 ;SHIFT 1	
Q656=+0 ;SHIFT 2	
7 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+20 ;PLUNGING DEPTH	
Q370=+0.424 ;TOOL PATH OVERLAP	
Q207=+6800 ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253= AUTO ;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	
Q576=+10000	;SPINDLE SPEED	
Q579=+0.7	;PLUNGING FACTOR S	
Q575=+1	;INFEEED STRATEGY	
8 L X+50 Y+50 R0 FMAX M99		Positioning and cycle call
9 TOOL CALL "MILL_D8_FINISH" Z S10000 F2000		Tool call, diameter 8 mm
10 L Z+250 R0 FMAX M3		
11 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= AUTO	;F PRE-POSITIONING	
Q200=+2	;SET-UP CLEARANCE	
Q438=-1	;ROUGH-OUT TOOL	
Q595=+1	;STRATEGY	
Q577=+0.2	;APPROACH RADIUS FACTOR	
12 L X+50 Y+50 R0 FMAX M99		Positioning and cycle call
13 CYCL DEF 274 OCM FINISHING SIDE		Define the side finishing cycle
Q338=+15	;INFEEED FOR FINISHING	
Q385= AUTO	;FINISHING FEED RATE	
Q253= AUTO	;F PRE-POSITIONING	
Q200=+2	;SET-UP CLEARANCE	
Q14=+0	;ALLOWANCE FOR SIDE	
QS438="MILL_D8"	;ROUGH-OUT TOOL	
Q351=+1	;CLIMB OR UP-CUT	
14 L X+50 Y+50 R0 FMAX M99		Positioning and cycle call
15 M30		End of program
16 END PGM OCM_FIGURE MM		

11

Cycles:
Cylinder surface

11.1 Fundamentals

Overview of cylindrical surface cycles

Soft key	Cycle	Page
	CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, option 8) <ul style="list-style-type: none"> ■ Milling of guide slots on the cylinder surface ■ Slot width is equal to tool radius 	339
	CYLINDER SURFACE slot milling (Cycle 28, DIN/ISO: G128, Option 8) <ul style="list-style-type: none"> ■ Milling of guide slots on the cylinder surface ■ Input of the slot width 	342
	CYLINDRICAL SURFACE ridge milling (Cycle 29, DIN/ISO: G129, Option 8) <ul style="list-style-type: none"> ■ Milling of a ridge on the cylinder surface ■ Input of the ridge width 	346
	CYLINDRICAL SURFACE CONTOUR (Cycle 39, DIN/ISO: G139, option 8) <ul style="list-style-type: none"> ■ Milling of a contour on the cylinder surface 	349

11.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, option 8)

Application



Refer to your machine manual!

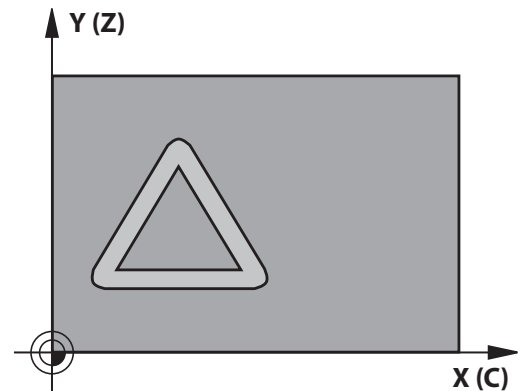
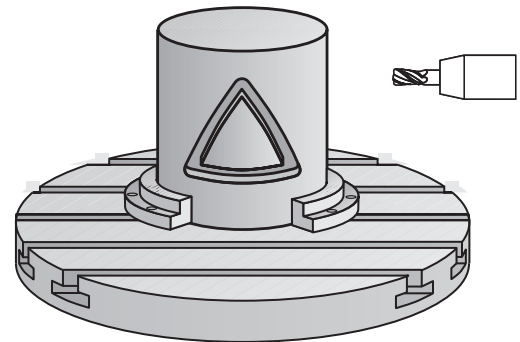
This function must be enabled and adapted by the machine tool builder.

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** to mill guide slots on the cylinder.

Describe the contour in a subprogram that you program with 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**.



Cycle run

- 1 The control positions the tool above the cutter infeed point, taking the finishing 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 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.



Operating note:

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

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



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

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 8)

Application



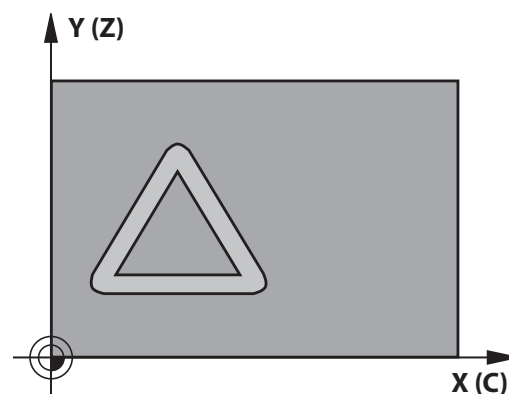
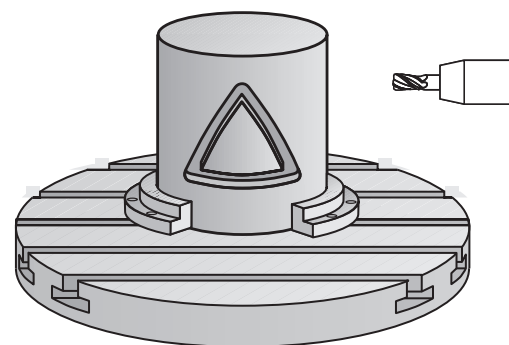
Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

With this cycle you can program a guide slot 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.



Cycle run

- 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.

**Operating notes:**

Define the approach in **apprDepCylWall** (no. 201004)

- CircleTangential:
Tangential approach and departure
- LineNormal: The tool approaches the contour starting point on a straight line
- The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Please note while programming:

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.

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



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

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.
Recommendation: Use a tolerance of 0.02 mm.
Function inactive: Enter 0 (default setting).
Input range for tolerance: 0.0001 to 9.9999

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 8)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

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.

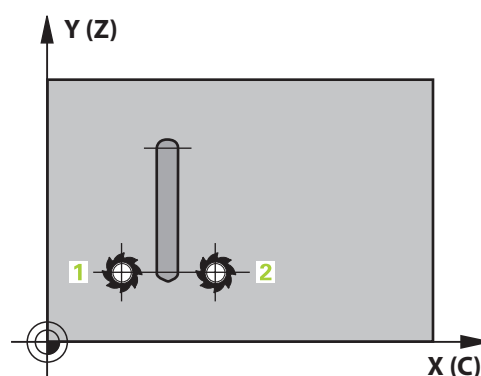
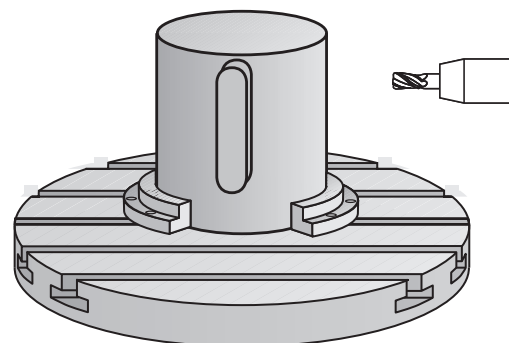
Cycle run

- 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 ridge 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.



Operating note:

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



Please note while programming:

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.

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 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 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 8)

Application



Refer to your machine manual!

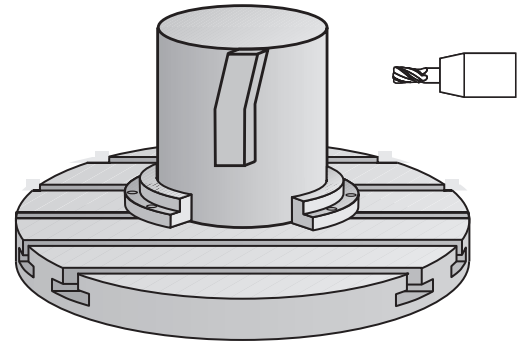
This function must be enabled and adapted by the machine tool builder.

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.

Describe the contour in a subprogram that you program with 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 contour actually to be machined.



Cycle run

- 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.

**Operating notes:**

Define the approach in **apprDepCylWall** (no. 201004)

- CircleTangential: Tangential approach and departure
- LineNormal: The tools approaches the contour starting point on a straight line
- The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Please note while programming:

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.

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



Ensure that the tool has enough space laterally for contour approach and departure.
The machining time can increase if the contour consists of many non-tangential contour elements.

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

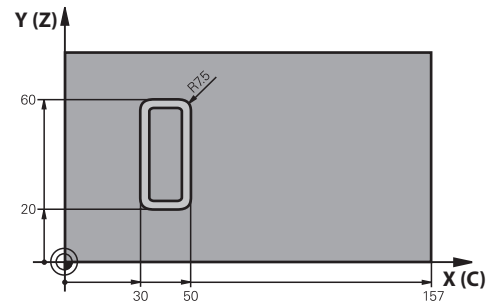
63 CYCL DEF 39 CYL. SURFACE CONTOUR	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION

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

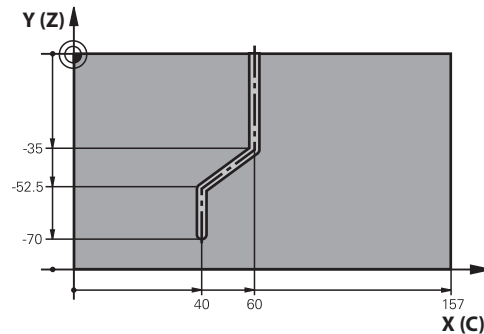


0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Positioning
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 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	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

**Cycles: Contour
pocket with
contour formula**

12.1 SL or OCM cycles with complex contour formula

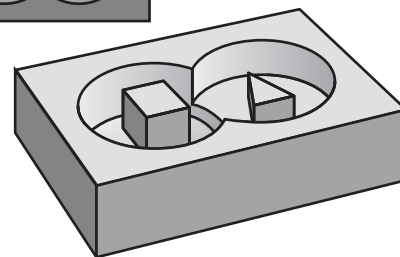
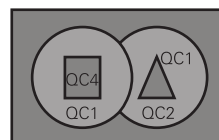
Fundamentals

Using complex contour formulas, you can combine several subcontours (pockets or islands) to program 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.



Programming notes:

- 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 CONTOUR DATA ...
8 CYCL DEF 22 ROUGH-OUT ...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM 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.

Cycle properties

- 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 dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM CONTOUR DATA**.

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
...
...

```

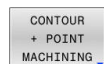
Selecting an NC program with contour definitions

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

or



- ▶ Press the **SELECT FILE** soft key and select the desired program
- ▶ Confirm your input with the **END** key







Programming notes:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
- 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:

- | | |
|--|---|
|  | <ul style="list-style-type: none"> ▶ Press the SPEC FCT key |
|  | <ul style="list-style-type: none"> ▶ Press the CONTOUR AND POINT MACHINING soft key |
|  | <ul style="list-style-type: none"> ▶ 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. |
| or | |
|  | <ul style="list-style-type: none"> ▶ Press the SELECT FILE soft key and select the desired NC program ▶ Define a separate depth for the selected contour ▶ Press the END key |



Programming notes:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
 - 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 or OCM cycles with simple contour formula", Page 369







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

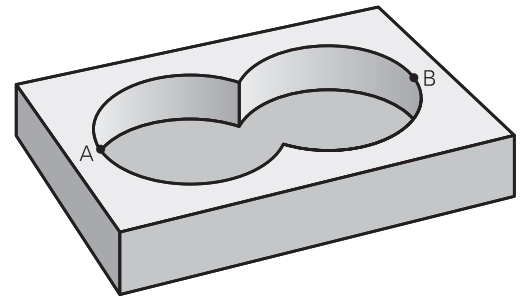
The control displays the following soft keys:

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. $QC12 = QC5 \wedge QC25$
	without e.g. $QC25 = QC1 \setminus QC2$
	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
```

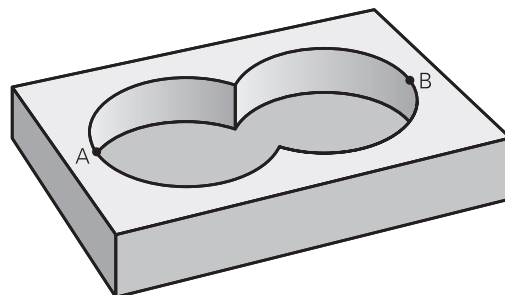
Contour description program 2: pocket B

```
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 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
```

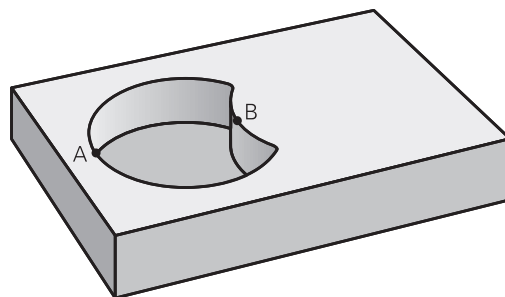
```
55 ...
```

```
56 ...
```

Area of exclusion

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

- Surfaces A and B must 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
```

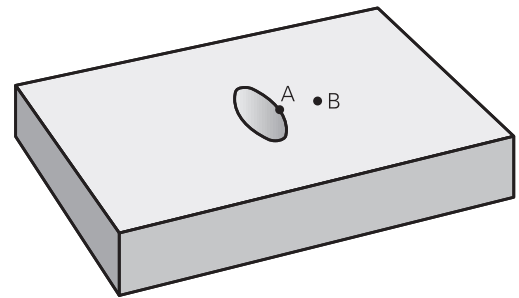
```
55 ...
```

```
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 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
```

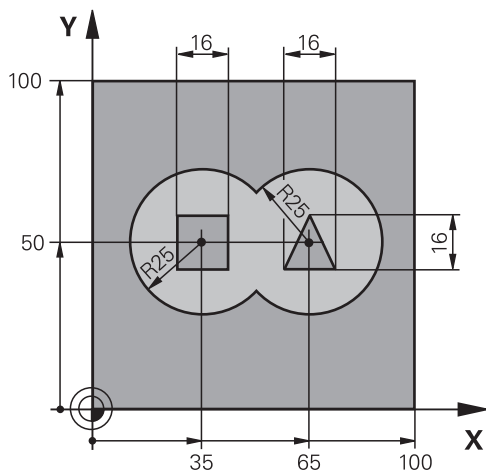
```
55 ...
```

```
56 ...
```

Machining contours with SL or OCM cycles

The entire contour is machined with the SL cycles (see "Overview", Page 244) or the OCM cycles (see "Overview", Page 289).

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 "CIRCLE31XY" 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 or OCM cycles with simple contour formula

Fundamentals

Using simple contour formulas, you can easily combine up to nine subcontours (pockets or islands) to program 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  ROUGH-OUT ...
9 CYCL CALL
...
12 CYCL DEF 23  FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24  SIDE FINISHING ...
17 CYCL CALL
63 L  Z+250 R0  FMAX M2
64 END PGM  CONTDEF MM

```

Properties of the subcontours

- Do not program a radius compensation.
- The 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.

Cycle properties






- 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, allowances and set-up clearance, can be entered centrally in Cycle **20 CONTOUR DATA**.







Entering a simple 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 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
- 
 - ▶ Press the **POCKET** soft key
- or
- 
 - ▶ 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
	Define the name of the contour or
	Press the SELECT FILE soft key
	Define the number of a string parameter
	Define the number of a label
	Define the name of the label
	Define the number of a string parameter for a label



Programming notes:

- The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth. Therefore, always start programming the subcontour 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 you enter a value of 0 for the depth, then the depth defined in Cycle **20** is effective for pockets. For islands, this means that they extend up to the workpiece surface!
- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Contour machining with SL Cycles



The entire contour is machined with the SL cycles (see "Overview", Page 244) or the OCM cycles (see "Overview", Page 289).










13

**Cycles: Special
Functions**

13.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	DWELL TIME (Cycle 9, ISO: G04) ■ Delay execution by the programmed dwell time	375
	PROGRAM CALL (Cycle 12, ISO: G39) ■ Call any NC program	376
	SPINDLE ORIENTATION (Cycle 13, ISO: G36) ■ Rotate spindle to a specific angle	377
	TOLERANCE (Cycle 32, DIN/ISO: G62) ■ Program the permissible contour deviation for jerk-free machining operations	378
	ENGRAVING (Cycle 225, ISO: G225) ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	381
	FACE MILLING (Cycle 232, DIN/ISO: G232, Option 19) ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	387
	MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155) ■ Determine the current machine status or test the measuring sequence	393
	ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143) ■ Selection for a weighing run ■ Reset the load-dependent feedforward and controller parameters	395
	THREAD CUTTING (Cycle 18, DIN/ISO: G86) ■ With controlled spindle ■ Spindle stops at the bottom of the hole	398

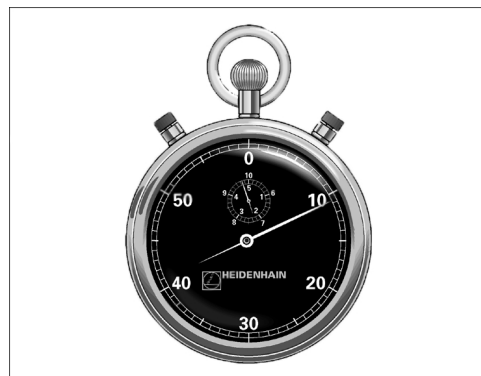
13.2 DWELL TIME (Cycle 9, ISO: G04)

Application

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** machining modes.



Example

89 CYCL DEF 9.0 DWELL TIME

90 CYCL DEF 9.1 DWELL 1.5

Cycle parameters

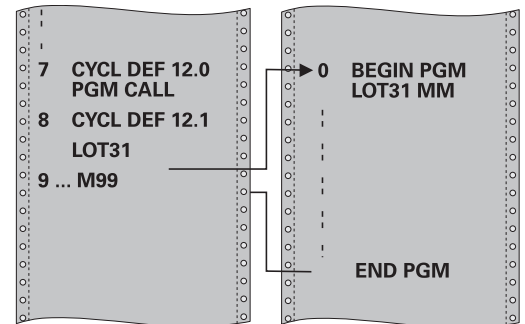


- **Dwell time in seconds:** Enter the dwell time in seconds.
Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

13.3 PROGRAM CALL (Cycle 12, ISO: G39)

Application

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** 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



- ▶ **Program name:** Enter the name of the NC program to be called 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)

Application



Refer to your machine manual!
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

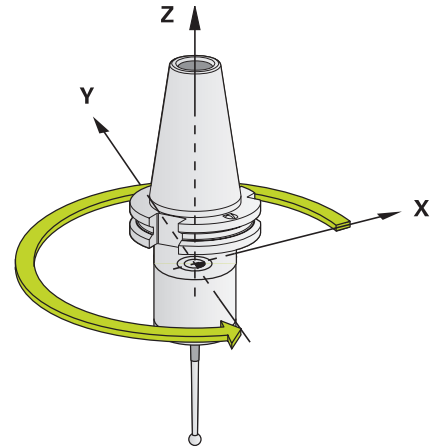
The control can control the main machine tool spindle and rotate it to a given angular position.

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** 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)

Application



Refer to your machine manual!
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, in as much as the control has been adapted to the machine's characteristics.

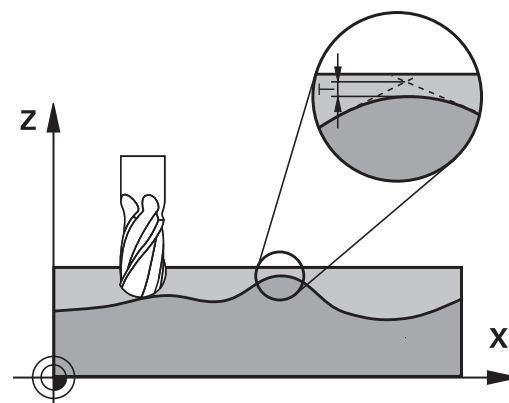
The control automatically smooths the contour between any two contour elements (whether compensated or not). This means that the tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular 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.



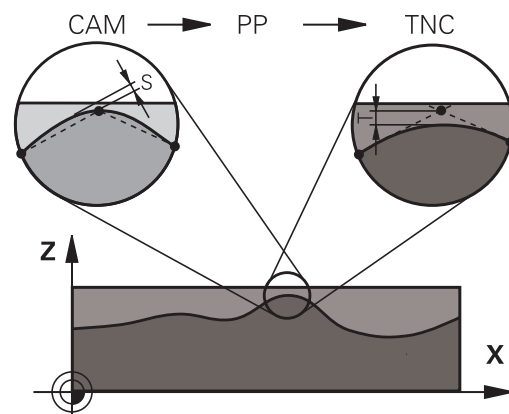
With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the control, but by the fact that, in order to machine the contour transitions very exactly, the control might have to drastically reduce the speed.



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 postprocessor (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 of the contour if you choose a tolerance value in Cycle **32** between 110 % and 200 % of the CAM chord error.



Please note while programming!

- This cycle can be executed in the **FUNCTION MODE MILL** machining modes.
- Cycle **32** is DEFactive 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 defined cycle parameters 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 uniform. In Cycle , you can additionally set a higher rotary axis tolerance **TA** (e.g., between 1° and 3°) for an even more constant feed-rate curve at the tool center 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:

$$T \sim K \times L \times TA \quad K = 0.0175 [1/^\circ]$$
 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.

$$T_w = \frac{180}{\pi \cdot R} T_{32}$$

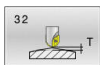
T_w : Angle tolerance in degrees

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).
 - ▶ **>0:** If you enter a value greater than zero, the control will use the maximum permissible deviation you specified.
 - ▶ **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.
Input range: 0.0000 to 10.0000
- ▶ **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 **Tool Center Point (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.
 - ▶ **>0:** If you enter a value greater than zero, the control will use the maximum permissible deviation you specified.
 - ▶ **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.
Input range: 0.0000 to 10.0000

Example

95 CYCL DEF 32.0 TOLERANCE

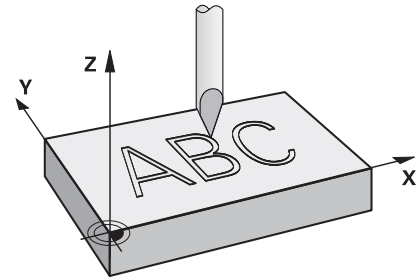
96 CYCL DEF 32.1 T0.05

97 CYCL DEF 32.2 HSC-MODE:1 TA5

13.6 ENGRAVING (Cycle 225, ISO: G225)

Application

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.



Cycle run

- 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 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 2nd set-up clearance.

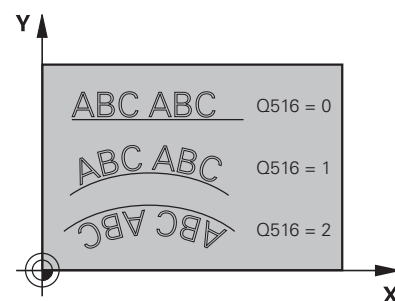
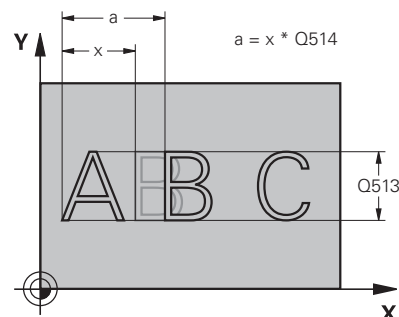
Please note while programming:

- 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 (**Q5**).
- 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



- ▶ **Q500 Engraving text?:** Text to be engraved, in quotation marks. 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 385
Maximum input: 255 characters
- ▶ **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)
Input range: -99999.9999 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**
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface.
Input range: 0 to 99999.9999; alternatively **PREDEF**



Example

62 CYCL DEF 225 ENGRAVING	
Q500=" " ;	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

- ▶ **Q203 Workpiece surface coordinate?** (absolute):
Coordinate of the workpiece surface with reference to the active preset
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**
- ▶ **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.
Input range: 0 to 999.9999

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. If 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, the current time, or the current calendar week. Do do so, enter **%time<x>**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY. (Identical to the **SYSSTR ID10321** 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
Calendar Week	%time99

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 use Cycle **225** to engrave the current counter reading (refer to the MOD menu).

To do so, program Cycle **225** as usual and enter the following as the text to be engraved, for example: **%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



Operating notes:

- 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.

13.7 FACE MILLING (Cycle 232, DIN/ISO: G232, Option 19)

Application



This function must be enabled and adapted by the machine tool builder.

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:

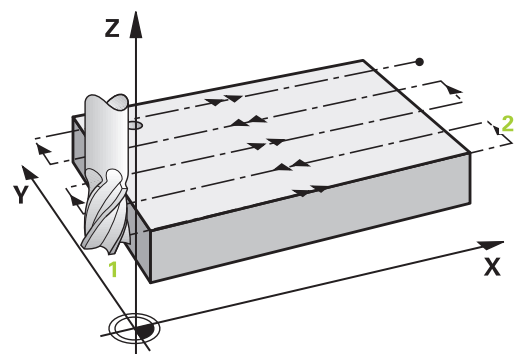
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1:** Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

Cycle run

- 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 working plane and then in the spindle axis. Otherwise, it first moves it to 2nd set-up clearance and then 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 tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

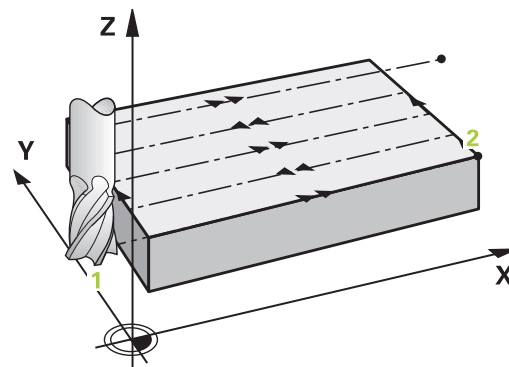
Strategy Q389=0

- 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.

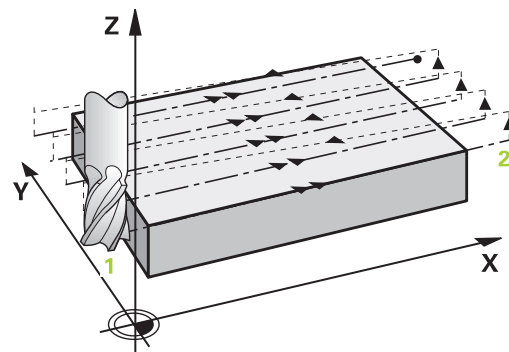


Strategy Q389=1

- 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.
- 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.

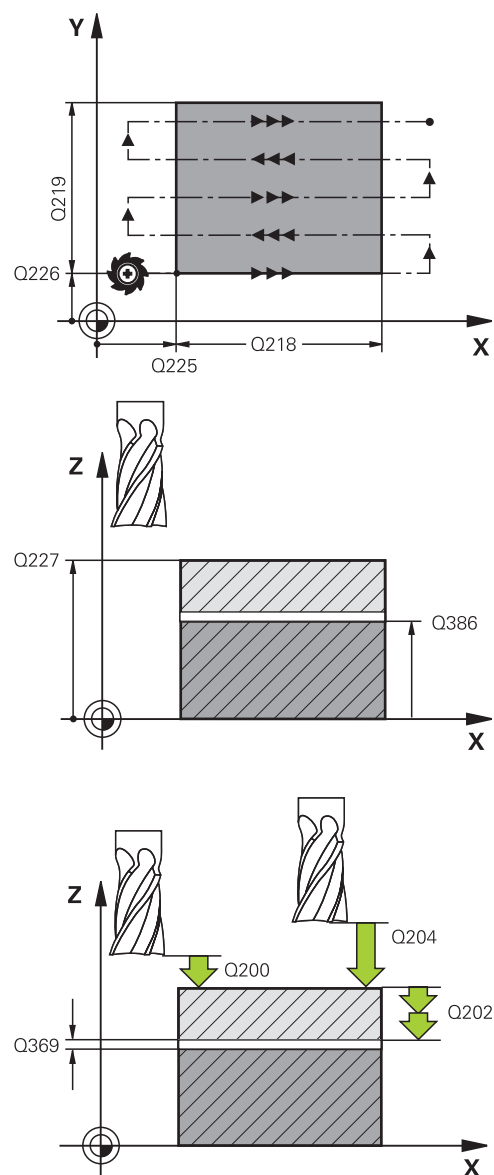


Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

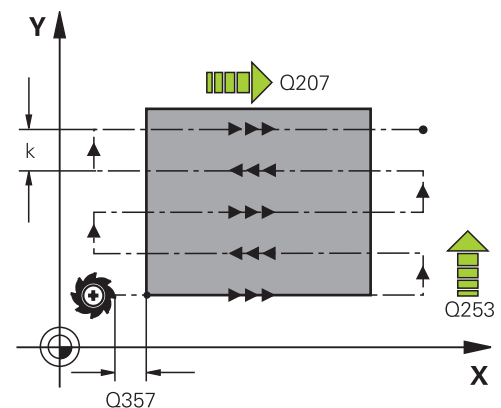
Cycle parameters



- ▶ **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 2nd 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 set-up
clearance above the current plunging depth to the
starting point of the next pass.
Input range: 0 to 99999.9999

**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

- ▶ **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**

13.8 MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)

Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

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. As a result, any deviations from the machine's shipping condition 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 and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

The machine tool builder can define warning and error thresholds for the measured values and optionally specify error reactions.

Cycle run



Operating note:

- Ensure that the axes are not clamped before you start the measurement.

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 task 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** machining modes.
- Cycle **238** is CALL-active.

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
```

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

Application

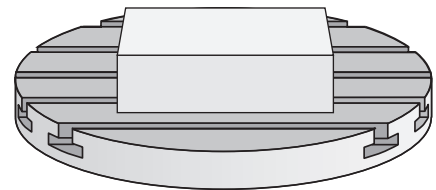


Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

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.



Cycle run**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.

**Operating note:**

- 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.

Please note while programming:**NOTICE****Danger of collision!**

The cycle can execute extensive movements in several axes at rapid traverse!

- ▶ Contact your machine tool builder to learn about the type and range of movements in Cycle **239** before using the cycle.
- ▶ 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** machining modes.
- Cycle **239** becomes effective immediately after its definition.
- 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

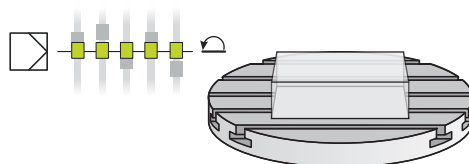


- **Q570 Load (0 = Delete/1 = Ascertain)?**: Specify whether the control should perform a LAC (Load Adaptive Control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters should be reset:

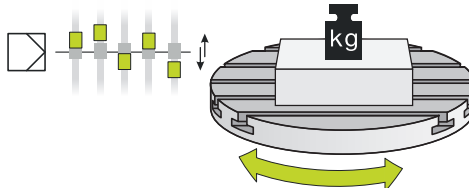
0: Reset LAC; the values most recently ascertained by the control are reset, and the control uses load-independent feedforward and controller parameters

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.

Q570 = 0



Q570 = 1



Example

62 CYCL DEF 239 ASCERTAIN THE
LOAD

Q570=+0 ;LOAD ASCERTATION

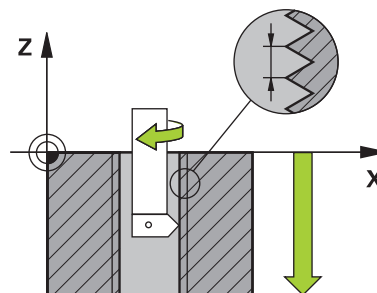
13.10 THREAD CUTTING (Cycle 18, DIN/ISO: G86)

Application



This function must be enabled and adapted by the machine tool builder.

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.



Operating note:

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)

Please note while programming:**NOTICE****Danger of collision!**

If you do not program a pre-positioning step before programming the call of Cycle **18**, a collision might occur. Cycle **18** does not perform any approach or departure movements.

- ▶ 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 the start of this cycle, Cycle **18** will switch it off and the cycle will execute with a stationary spindle! At the end, Cycle **18** will switch the spindle on again if it was on before the start of the cycle.

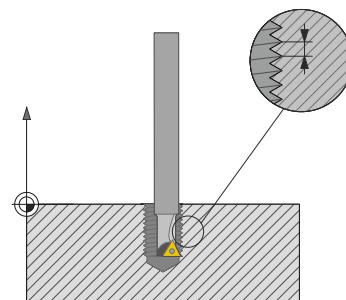
- ▶ Before starting this cycle, be sure to program a spindle stop! (For example with **M5**)
- ▶ At the end of Cycle **18**, the control restores the spindle to its state at cycle start. This means that if the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle **18**.

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

14

Tables of Cycles

14.1 Table of cycles



All cycles that are not related to the machining cycles are described in the **Programming of Measuring Cycles for Workpieces and Tools** User's Manual. This manual is available from HEIDENHAIN upon request.
ID of User's Manual for Programming of Measuring Cycles for Workpieces and Tools: 1303431-xx

Machining cycles

Cycle number	Cycle name	DEF active	CALL active	Page
7	DATUM SHIFT	■		203
8	MIRRORING	■		210
9	DWELL TIME	■		375
10	ROTATION	■		211
11	SCALING FACTOR	■		213
12	PGM CALL	■		376
13	ORIENTATION	■		377
14	CONTOUR	■		245
18	THREAD CUTTING		■	398
19	WORKING PLANE	■		216
20	CONTOUR DATA	■		250
21	PILOT DRILLING		■	252
22	ROUGH-OUT		■	254
23	FLOOR FINISHING		■	258
24	SIDE FINISHING		■	260
25	CONTOUR TRAIN		■	265
26	AXIS-SPECIFIC SCALING	■		214
27	CYLINDER SURFACE		■	339
28	CYLINDER SURFACE		■	342
29	CYL SURFACE RIDGE		■	346
32	TOLERANCE	■		378
39	CYL. SURFACE CONTOUR		■	349
200	DRILLING		■	72
201	REAMING		■	74
202	BORING		■	76
203	UNIVERSAL DRILLING		■	80
204	BACK BORING		■	85
205	UNIVERSAL PECKING		■	89
206	TAPPING		■	113

Cycle number	Cycle name	DEF active	CALL active	Page
207	RIGID TAPPING		■	116
208	BORE MILLING		■	95
209	TAPPING W/ CHIP BRKG		■	120
220	POLAR PATTERN	■		230
221	CARTESIAN PATTERN	■		233
224	DATAMATRIX CODE PATTERN	■		236
225	ENGRAVING		■	381
232	FACE MILLING		■	387
233	FACE MILLING (milling direction can be selected, take the side walls into account)		■	190
238	MEASURE MACHINE STATUS	■		393
239	ASCERTAIN THE LOAD	■		395
240	CENTERING		■	106
241	SINGLE-LIP D.H.DRLNG		■	98
247	PRESETTING	■		223
251	RECTANGULAR POCKET		■	151
252	CIRCULAR POCKET		■	158
253	SLOT MILLING		■	165
254	CIRCULAR SLOT		■	169
256	RECTANGULAR STUD		■	175
257	CIRCULAR STUD		■	180
258	POLYGON STUD		■	184
262	THREAD MILLING		■	126
263	THREAD MLLNG/CNTSNKG		■	130
264	THREAD DRILLNG/MLLNG		■	134
265	HEL. THREAD DRLG/MLG		■	138
267	OUTSIDE THREAD MLLNG		■	142
270	CONTOUR TRAIN DATA		■	263
271	OCM CONTOUR DATA		■	290
272	OCM ROUGHING		■	292
273	OCM FINISHING FLOOR		■	304
274	OCM FINISHING SIDE		■	307
275	TROCHOIDAL SLOT		■	269
276	THREE-D CONT. TRAIN		■	274
277	OCM CHAMFERING		■	309
1271	OCM RECTANGLE	■		313
1272	OCM CIRCLE	■		316
1273	OCM SLOT / RIDGE	■		318

Cycle number	Cycle name	DEF active	CALL active	Page
1278	OCM POLYGON	■		320
1281	OCM RECTANGLE BOUNDARY	■		323
1282	OCM CIRCLE BOUNDARY	■		325

Index

2

2-D CODE..... 236

A

About this manual..... 28
Ascertain the load..... 395

C

Circular hole pattern..... 230
Contour cycles..... 242
Coordinate transformation
 Axis-specific scaling factor... 214
 Datum shift..... 203, 204
 Fundamentals..... 202
 Mirroring..... 210
 Rotation..... 211
 Scaling..... 213
Cycle..... 46
 Calling..... 48
 Define..... 47
Cycles and point tables..... 66
Cylinder surface cycles
 Contour..... 349
 Cylinder surface..... 339
 Fundamentals..... 338
 Ridge..... 346
 Slot..... 342

D

Datum shift
 Programming..... 203
 With datum tables..... 204
Drilling Cycles..... 70
 Back boring..... 85
 Bore milling..... 95
 Boring..... 76
 Centering..... 106
 Drilling..... 72
 Reaming..... 74
 Single-lip deep-hole drilling... 98
 Universal drilling..... 80
 Universal pecking..... 89
Dwell time..... 375

E

Engraving..... 381

F

Face milling..... 190, 387
Feature content level..... 34
Floor finishing..... 258

G

GLOBAL DEF..... 52

M

Machining patterns..... 57
Measure machine status..... 393

O

OCM
 Chamfering..... 309
 Contour data..... 290
 Cutting data calculator..... 296
 Finishing floor..... 304
 Finishing side..... 307
 Roughing..... 292
 Standard shapes..... 312
OCM cycles..... 286
 With complex contour
 formula..... 358
 With simple contour formula... 369
OCM shapes
 Circle..... 316
 Circular boundary..... 325
 Polygon..... 320
 Rectangle..... 313
 Rectangular boundary..... 323
 Slot/ridge..... 318
Option..... 31

P

Pattern
 Circle..... 230
 DataMatrix code..... 236
 Lines..... 233
PATTERN DEF
 entering..... 58
 using..... 58
Pattern definition with PATTERN
DEF..... 57
 frames..... 61
 full circle..... 62
 patterns..... 60
 pitch circle..... 63
 Point..... 59
Pecking..... 89
Pocket milling cycles
 Circular pocket..... 158
 Rectangular pocket..... 151
Point patterns..... 228
Point tables..... 64
Presetting..... 223
Program call
 Via cycle..... 376

S

Side finishing..... 260
SL Cycles..... 242
 3-D contour train..... 274
 Contour..... 245
 Contour data..... 250

Contour train..... 265
Contour train data..... 263
Floor finishing..... 258
Fundamentals..... 242
OCM Chamfering..... 309
OCM contour data..... 290
OCM Finishing floor..... 304
OCM Finishing side..... 307
OCM fundamentals..... 286
OCM roughing..... 292
Pilot drilling..... 252
Roughing..... 254
Side finishing..... 260
Superimposed contours.... 246, 363
Trochoidal Slot..... 269
 With complex contour
 formula..... 358
 With simple contour formula... 369
Slot milling cycles
 Circular slot..... 169
 Slot milling..... 165
Software option..... 31
Spindle orientation..... 377
Stud milling cycles
 Circular stud..... 180
 Polygon stud..... 184
 Rectangular stud..... 175

T

Table of cycles..... 402
 Machining cycles..... 402
Tapping..... 112
 Rigid tapping..... 116
 with a floating tap holder.... 113
 with chip breaking..... 120
Thread cutting..... 398
Thread milling
 Fundamentals..... 124
 Helical thread drilling/milling 138
 internal threads..... 126
 outside..... 142
 thread drilling/milling..... 134
 Thread milling/countersinking... 130
Tilt working plane
 Procedure..... 222
Tolerance..... 378

W

Working plane..... 216

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.

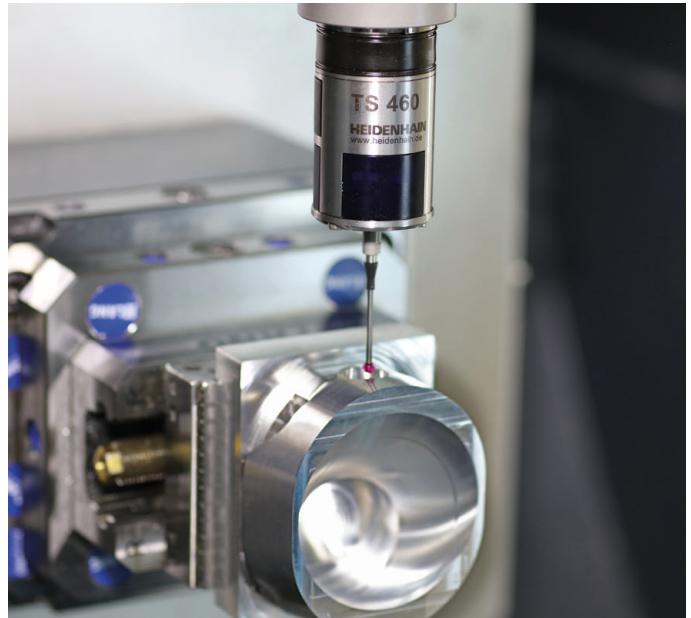
Workpiece touch probes

TS 248, TS 260 Signal transmission by cable

TS 460 Radio or infrared transmission

TS 640, 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

