



# HEIDENHAIN



## TNC 620

User's Manual  
Programming of  
Machining Cycles

NC Software  
817600-16  
817601-16  
817605-16

English (en)  
01/2022



## Contents

<b>1</b>	<b>Fundamentals.....</b>	<b>21</b>
<b>2</b>	<b>Fundamentals / Overviews.....</b>	<b>35</b>
<b>3</b>	<b>Using Fixed Cycles.....</b>	<b>39</b>
<b>4</b>	<b>Cycles: Drilling.....</b>	<b>67</b>
<b>5</b>	<b>Cycles: Tapping / Thread Milling.....</b>	<b>119</b>
<b>6</b>	<b>Cycles: Pocket Milling / Stud Milling / Slot Milling.....</b>	<b>161</b>
<b>7</b>	<b>Cycles: Coordinate Transformations.....</b>	<b>221</b>
<b>8</b>	<b>Cycles: Pattern Definitions.....</b>	<b>241</b>
<b>9</b>	<b>Cycles: Contour Pocket.....</b>	<b>259</b>
<b>10</b>	<b>Cycles: Optimized Contour Milling.....</b>	<b>307</b>
<b>11</b>	<b>Cycles: Cylinder Surface.....</b>	<b>369</b>
<b>12</b>	<b>Cycles: Contour Pocket with Contour Formula.....</b>	<b>389</b>
<b>13</b>	<b>Cycles: Special Functions.....</b>	<b>405</b>
<b>14</b>	<b>Tables of Cycles.....</b>	<b>435</b>



<b>1</b>	<b>Fundamentals.....</b>	<b>21</b>
1.1	About this manual.....	22
1.2	Control model, software, and features.....	24
	Software options.....	25
	New and modified cycle functions of software 81760x-16.....	30

<b>2</b>	<b>Fundamentals / Overviews.....</b>	<b>35</b>
2.1	Introduction.....	36
2.2	Available cycle groups.....	37
	Overview of fixed cycles.....	37
	Overview of touch probe cycles.....	38

<b>3</b>	<b>Using Fixed Cycles.....</b>	<b>39</b>
<b>3.1</b>	<b>Working with fixed cycles.....</b>	<b>40</b>
	Machine-specific cycles (option 19).....	40
	Defining a cycle using soft keys.....	41
	Defining a cycle using the GOTO function.....	42
	Calling a cycle.....	43
<b>3.2</b>	<b>Program defaults for cycles.....</b>	<b>47</b>
	Overview.....	47
	Entering GLOBAL DEF.....	47
	Using GLOBAL DEF information.....	48
	Global data valid everywhere.....	49
	Global data for drilling operations.....	50
	Global data for milling operations with pocket cycles.....	51
	Global data for milling operations with contour cycles.....	52
	Global data for positioning behavior.....	52
	Global data for probing functions.....	53
<b>3.3</b>	<b>Pattern definition with PATTERN DEF.....</b>	<b>54</b>
	Application.....	54
	Entering PATTERN DEF.....	55
	Using PATTERN DEF.....	55
	Defining individual machining positions.....	56
	Defining a single row.....	57
	Defining an individual pattern.....	58
	Defining an individual frame.....	60
	Defining a full circle.....	62
	Defining a pitch circle.....	63
<b>3.4</b>	<b>Point tables with cycles.....</b>	<b>64</b>
	Application with cycles.....	64
	Calling a cycle in connection with point tables.....	64

<b>4</b>	<b>Cycles: Drilling.....</b>	<b>67</b>
<b>4.1</b>	<b>Fundamentals.....</b>	<b>68</b>
	Overview.....	68
<b>4.2</b>	<b>Cycle 200 DRILLING.....</b>	<b>70</b>
	Cycle parameters.....	72
<b>4.3</b>	<b>Cycle 201 REAMING (option 19).....</b>	<b>74</b>
	Cycle parameters.....	75
<b>4.4</b>	<b>Cycle 202 BORING (option 19).....</b>	<b>76</b>
	Cycle parameters.....	78
<b>4.5</b>	<b>Cycle 203 UNIVERSAL DRILLING (option 19).....</b>	<b>80</b>
	Cycle parameters.....	83
<b>4.6</b>	<b>Cycle 204 BACK BORING (option 19).....</b>	<b>85</b>
	Cycle parameters.....	87
<b>4.7</b>	<b>Cycle 205 UNIVERSAL PECKING (option 19).....</b>	<b>89</b>
	Cycle parameters.....	92
	Chip removal and chip breaking.....	95
<b>4.8</b>	<b>Cycle 208 BORE MILLING (option 19).....</b>	<b>97</b>
	Cycle parameters.....	100
<b>4.9</b>	<b>Cycle 241 SINGLE-LIP D.H.DRLNG (option 19).....</b>	<b>102</b>
	Cycle parameters.....	104
	User macro.....	107
	Position behavior when working with Q379.....	108
<b>4.10</b>	<b>Cycle 240 CENTERING (option 19).....</b>	<b>112</b>
	Cycle parameters.....	114
<b>4.11</b>	<b>Programming Examples.....</b>	<b>116</b>
	Example: Drilling cycles.....	116
	Example: Using cycles in connection with PATTERN DEF.....	117



<b>5</b>	<b>Cycles: Tapping / Thread Milling.....</b>	<b>119</b>
<b>5.1</b>	<b>Fundamentals.....</b>	<b>120</b>
	Overview.....	120
<b>5.2</b>	<b>Cycle 206 TAPPING.....</b>	<b>121</b>
	Cycle parameters.....	123
<b>5.3</b>	<b>Cycle 207 RIGID TAPPING.....</b>	<b>124</b>
	Cycle parameters.....	126
	Retracting after a program interruption.....	127
<b>5.4</b>	<b>Cycle 209 TAPPING W/ CHIP BRKG (option 19).....</b>	<b>128</b>
	Cycle parameters.....	130
	Retracting after a program interruption.....	132
<b>5.5</b>	<b>Fundamentals of thread milling.....</b>	<b>133</b>
	Requirements.....	133
<b>5.6</b>	<b>Cycle 262 THREAD MILLING (option 19).....</b>	<b>135</b>
	Cycle parameters.....	137
<b>5.7</b>	<b>Cycle 263 THREAD MLLNG/CNTSNKG (option 19).....</b>	<b>139</b>
	Cycle parameters.....	141
<b>5.8</b>	<b>Cycle 264 THREAD DRILLNG/MLLNG (option 19).....</b>	<b>144</b>
	Cycle parameters.....	146
<b>5.9</b>	<b>Cycle 265 HEL. THREAD DRLG/MLG (option 19).....</b>	<b>149</b>
	Cycle parameters.....	151
<b>5.10</b>	<b>Cycle 267 OUTSIDE THREAD MLLNG (option 19).....</b>	<b>153</b>
	Cycle parameters.....	155
<b>5.11</b>	<b>Programming Examples.....</b>	<b>158</b>
	Example: Thread milling.....	158

<b>6</b>	<b>Cycles: Pocket Milling / Stud Milling / Slot Milling.....</b>	<b>161</b>
<b>6.1</b>	<b>Fundamentals.....</b>	<b>162</b>
	Overview.....	162
<b>6.2</b>	<b>Cycle 251 RECTANGULAR POCKET (option 19).....</b>	<b>163</b>
	Cycle parameters.....	165
	Plunging strategy Q366 with RCUTS.....	169
<b>6.3</b>	<b>Cycle 252 CIRCULAR POCKET (option 19).....</b>	<b>170</b>
	Cycle parameters.....	173
	Plunging strategy Q366 with RCUTS.....	176
<b>6.4</b>	<b>Cycle 253 SLOT MILLING (option 19).....</b>	<b>177</b>
	Cycle parameters.....	179
<b>6.5</b>	<b>Cycle 254 CIRCULAR SLOT (option 19).....</b>	<b>183</b>
	Cycle parameters.....	185
<b>6.6</b>	<b>Cycle 256 RECTANGULAR STUD (option 19).....</b>	<b>189</b>
	Cycle parameters.....	191
<b>6.7</b>	<b>Cycle 257 CIRCULAR STUD (option 19).....</b>	<b>195</b>
	Cycle parameters.....	197
<b>6.8</b>	<b>Cycle 258 POLYGON STUD (option 19).....</b>	<b>200</b>
	Cycle parameters.....	202
<b>6.9</b>	<b>Cycle 233 FACE MILLING (option 19).....</b>	<b>206</b>
	Cycle parameters.....	212
<b>6.10</b>	<b>Programming Examples.....</b>	<b>217</b>
	Example: Milling pockets, studs and slots.....	217

<b>7</b>	<b>Cycles: Coordinate Transformations.....</b>	<b>221</b>
<b>7.1</b>	<b>Fundamentals.....</b>	<b>222</b>
	Overview.....	222
	Effectiveness of coordinate transformations.....	222
<b>7.2</b>	<b>Cycle 7 DATUM SHIFT.....</b>	<b>223</b>
	Cycle parameters.....	225
<b>7.3</b>	<b>Cycle 8 MIRRORING.....</b>	<b>226</b>
	Cycle parameters.....	226
<b>7.4</b>	<b>Cycle 10 ROTATION.....</b>	<b>227</b>
	Cycle parameters.....	228
<b>7.5</b>	<b>Cycle 11 SCALING FACTOR.....</b>	<b>229</b>
	Cycle parameters.....	229
<b>7.6</b>	<b>Cycle 26 AXIS-SPECIFIC SCALING.....</b>	<b>230</b>
	Cycle parameters.....	230
<b>7.7</b>	<b>Cycle 19 WORKING PLANE (option 8).....</b>	<b>231</b>
	Cycle parameters.....	233
	Reset.....	233
	Positioning the axes of rotation.....	233
	Position display in a tilted system.....	235
	Monitoring of the working space.....	235
	Positioning in a tilted coordinate system.....	235
	Combining coordinate transformation cycles.....	235
	Procedure for working with Cycle 19 WORKING PLANE.....	236
<b>7.8</b>	<b>Cycle 247 PRESETTING.....</b>	<b>237</b>
	Cycle parameters.....	237
<b>7.9</b>	<b>Programming Examples.....</b>	<b>238</b>
	Example: coordinate transformation cycles.....	238

<b>8</b>	<b>Cycles: Pattern Definitions.....</b>	<b>241</b>
<b>8.1</b>	<b>Fundamentals.....</b>	<b>242</b>
	Overview.....	242
<b>8.2</b>	<b>Cycle 220 POLAR PATTERN (option 19).....</b>	<b>244</b>
	Cycle parameters.....	245
<b>8.3</b>	<b>Cycle 221 CARTESIAN PATTERN (option 19).....</b>	<b>247</b>
	Cycle parameters.....	249
<b>8.4</b>	<b>Cycle 224 DATAMATRIX CODE PATTERN (option 19).....</b>	<b>251</b>
	Cycle parameters.....	253
	Outputting variable texts in DataMatrix codes.....	254
<b>8.5</b>	<b>Programming Examples.....</b>	<b>257</b>
	Example: Polar hole patterns.....	257

<b>9</b>	<b>Cycles: Contour Pocket.....</b>	<b>259</b>
<b>9.1</b>	<b>SL Cycles.....</b>	<b>260</b>
	General information.....	260
	Overview.....	262
<b>9.2</b>	<b>Cycle 14 CONTOUR.....</b>	<b>263</b>
	Cycle parameters.....	263
<b>9.3</b>	<b>Superimposed contours.....</b>	<b>264</b>
	Fundamentals.....	264
	Subprograms: overlapping pockets.....	264
	Surface resulting from sum.....	265
	Surface resulting from difference.....	266
	Surface resulting from intersection.....	266
<b>9.4</b>	<b>Cycle 20 CONTOUR DATA (option 19).....</b>	<b>267</b>
	Cycle parameters.....	268
<b>9.5</b>	<b>Cycle 21 PILOT DRILLING (option 19).....</b>	<b>270</b>
	Cycle parameters.....	271
<b>9.6</b>	<b>Cycle 22 ROUGH-OUT (option 19).....</b>	<b>272</b>
	Cycle parameters.....	275
<b>9.7</b>	<b>Cycle 23 FLOOR FINISHING (option 19).....</b>	<b>277</b>
	Cycle parameters.....	279
<b>9.8</b>	<b>Cycle 24 SIDE FINISHING (option 19).....</b>	<b>280</b>
	Cycle parameters.....	282
<b>9.9</b>	<b>Cycle 270 CONTOUR TRAIN DATA (option 19).....</b>	<b>283</b>
	Cycle parameters.....	284
<b>9.10</b>	<b>Cycle 25 CONTOUR TRAIN (option 19).....</b>	<b>285</b>
	Cycle parameters.....	287
<b>9.11</b>	<b>Cycle 275 TROCHOIDAL SLOT (option 19).....</b>	<b>289</b>
	Cycle parameters.....	292
<b>9.12</b>	<b>Cycle 276 THREE-D CONT. TRAIN (option 19).....</b>	<b>295</b>
	Cycle parameters.....	298
<b>9.13</b>	<b>Programming examples.....</b>	<b>300</b>
	Example: Roughing-out and fine-roughing a pocket with SL Cycles.....	300
	Example: Pilot drilling, roughing and finishing overlapping contours with SL Cycles.....	302
	Example: Contour train.....	304

<b>10 Cycles: Optimized Contour Milling.....</b>	<b>307</b>
<b>10.1 OCM cycles (option 167).....</b>	<b>308</b>
OCM cycles.....	308
Overview.....	311
<b>10.2 Cycle 271 OCM CONTOUR DATA (option 167).....</b>	<b>312</b>
Cycle parameters.....	313
<b>10.3 Cycle 272 OCM ROUGHING (option 167).....</b>	<b>315</b>
Cycle parameters.....	318
<b>10.4 OCM cutting data calculator (option 167).....</b>	<b>321</b>
Fundamentals of the OCM cutting data calculator.....	321
Operation.....	323
Fillable form.....	323
Process parameters.....	327
Achieving an optimum result.....	328
<b>10.5 Cycle 273 OCM FINISHING FLOOR (option 167).....</b>	<b>330</b>
Cycle parameters.....	332
<b>10.6 Cycle 274 OCM FINISHING SIDE (option 167).....</b>	<b>334</b>
Cycle parameters.....	335
<b>10.7 Cycle 277 OCM CHAMFERING (option 167).....</b>	<b>337</b>
Cycle parameters.....	339
<b>10.8 OCM standard figures.....</b>	<b>341</b>
Fundamentals.....	341
<b>10.9 Cycle 1271 OCM RECTANGLE (option 167).....</b>	<b>343</b>
Cycle parameters.....	344
<b>10.10 Cycle 1272 OCM CIRCLE (option 167).....</b>	<b>346</b>
Cycle parameters.....	347
<b>10.11 Cycle 1273 OCM SLOT / RIDGE (option 167).....</b>	<b>349</b>
Cycle parameters.....	350
<b>10.12 Cycle 1278 OCM POLYGON (option 167).....</b>	<b>352</b>
Cycle parameters.....	353
<b>10.13 Cycle 1281 OCM RECTANGLE BOUNDARY (option 167).....</b>	<b>355</b>
Cycle parameters.....	356

<b>10.14 Cycle 1282 OCM CIRCLE BOUNDARY (option 167).....</b>	<b>357</b>
Cycle parameters.....	358
<b>10.15 Programming examples.....</b>	<b>359</b>
Example: Open pocket and fine roughing with OCM cycles.....	359
Example: Program various depths with OCM cycles.....	362
Example: Face milling and fine roughing with OCM cycles.....	364
Example: Contour with OCM figure cycles.....	366

<b>11 Cycles: Cylinder Surface.....</b>	<b>369</b>
<b>11.1 Fundamentals.....</b>	<b>370</b>
Overview of cylindrical surface cycles.....	370
<b>11.2 Cycle 27 CYLINDER SURFACE (option 8).....</b>	<b>371</b>
Cycle parameters.....	373
<b>11.3 Cycle 28 CYLINDRICAL SURFACE SLOT (option 8).....</b>	<b>374</b>
Cycle parameters.....	377
<b>11.4 Cycle 29 CYL SURFACE RIDGE (option 8).....</b>	<b>379</b>
Cycle parameters.....	381
<b>11.5 Cycle 39 CYL. SURFACE CONTOUR (option 8).....</b>	<b>383</b>
Cycle parameters.....	385
<b>11.6 Programming Examples.....</b>	<b>386</b>
Example: Cylinder surface with Cycle 27.....	386
Example: Cylinder surface with Cycle 28.....	388



<b>12 Cycles: Contour Pocket with Contour Formula.....</b>	<b>389</b>
<b>12.1 SL or OCM cycles with complex contour formula.....</b>	<b>390</b>
Fundamentals.....	390
Selecting an NC program with contour definitions.....	392
Defining contour descriptions.....	393
Entering a complex contour formula.....	394
Superimposed contours.....	395
Machining contours with SL or OCM cycles.....	397
Example: Roughing and finishing superimposed contours with the contour formula.....	397
<b>12.2 SL or OCM cycles with simple contour formula.....</b>	<b>400</b>
Fundamentals.....	400
Entering a simple contour formula.....	402
Contour machining with SL Cycles.....	403

<b>13 Cycles: Special Functions.....</b>	<b>405</b>
<b>13.1 Fundamentals.....</b>	<b>406</b>
Overview.....	406
<b>13.2 Cycle 9 DWELL TIME.....</b>	<b>407</b>
Cycle parameters.....	407
<b>13.3 Cycle 12 PGM CALL.....</b>	<b>408</b>
Cycle parameters.....	409
<b>13.4 Cycle 13 ORIENTATION.....</b>	<b>410</b>
Cycle parameters.....	410
<b>13.5 Cycle 32 TOLERANCE.....</b>	<b>411</b>
Influences of the geometry definition in the CAM system.....	412
Cycle parameters.....	414
<b>13.6 Cycle 225 ENGRAVING.....</b>	<b>415</b>
Cycle parameters.....	416
Allowed engraving characters.....	419
Characters that cannot be printed.....	419
Engraving system variables.....	420
Engraving the name and path of an NC program.....	421
Engraving the counter reading.....	421
<b>13.7 Cycle 232 FACE MILLING (option 19).....</b>	<b>422</b>
Cycle parameters.....	425
<b>13.8 Cycle 238 MEASURE MACHINE STATUS (option 155).....</b>	<b>428</b>
Cycle parameters.....	429
<b>13.9 Cycle 239 ASCERTAIN THE LOAD (option 143).....</b>	<b>430</b>
Cycle parameters.....	432
<b>13.10 Cycle 18 THREAD CUTTING.....</b>	<b>433</b>
Cycle parameters.....	434

<b>14 Tables of Cycles.....</b>	<b>435</b>
<b>14.1 Table of cycles.....</b>	<b>436</b>
Machining cycles.....	436



# 1

## Fundamentals

## 1.1 About this manual

### Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer'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 contain 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 manufacturer. 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 manufacturer 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	817600-16
TNC 620 E	817601-16
TNC 620 Programming station	817605-16

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.



### User's Manual:

All cycle functions not connected 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



### 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.

Klartext 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 manufacturer. The respective options provide the functions listed below:

---

### Additional Axis (option 0 and option 1)

---

<b>Additional axis</b>	Additional control loops 1 and 2
------------------------	----------------------------------

---

### Advanced Function Set 1 (option 8)

---

<b>Advanced functions (set 1)</b>	<b>Machining with rotary tables</b> <ul style="list-style-type: none"> <li>■ Cylindrical contours as if in two axes</li> <li>■ Feed rate in distance per minute</li> </ul> <b>Coordinate conversions:</b> Tilting the working plane
-----------------------------------	--

---

### Advanced Function Set 2 (option 9)

---

<b>Advanced functions (set 2)</b> Export license required	<b>3-D machining:</b> <ul style="list-style-type: none"> <li>■ 3-D tool compensation through surface-normal vectors</li> <li>■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = <b>T</b>ool <b>C</b>enter <b>P</b>oint <b>M</b>anagement)</li> <li>■ Keeping the tool normal to the contour</li> <li>■ Tool radius compensation normal to the tool direction</li> <li>■ Manual traverse in the active tool-axis system</li> </ul> <b>Interpolation:</b> Linear in > 4 axes (export license required)
--	---

---

### Touch Probe Functions (option 17)

---

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

---

### HEIDENHAIN DNC (option 18)

---

Communication with external PC applications over COM component

---

### Advanced Programming Features (option 19)

---

<b>Expanded programming functions</b>	<b>FK free contour programming:</b> 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)**

<b>Extended tool management</b>	Python-based expansion of tool management <ul style="list-style-type: none"> <li>■ Program-specific or pallet-specific usage sequence of all tools</li> <li>■ Program-specific or pallet-specific tooling list of all tools</li> </ul>
---------------------------------	--

**Remote Desktop Manager (option 133)**

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

**Cross Talk Compensation – CTC (option 141)**

<b>Compensation of axis couplings</b>	<ul style="list-style-type: none"> <li>■ Determination of dynamically caused position deviation through axis acceleration</li> <li>■ Compensation of the TCP (<b>T</b>ool <b>C</b>enter <b>P</b>oint)</li> </ul>
---------------------------------------	--

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

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

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

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

**Active Chatter Control – ACC (option 145)**

<b>Active chatter control</b>	Fully automatic function for chatter control during machining
-------------------------------	---

**Machine Vibration Control – MVC (option 146)**

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

**CAD Model Optimizer (option 152)**

<b>Optimization of CAD models</b>	Convert and optimize CAD models <ul style="list-style-type: none"> <li>■ Fixtures</li> <li>■ Workpiece blank</li> <li>■ Finished part</li> </ul>
-----------------------------------	--

**Batch Process Manager (option 154)**

<b>Batch process manager</b>	Planning of production orders
------------------------------	-------------------------------

**Component Monitoring (option 155)**

<b>Component monitoring without external sensors</b>	Monitoring configured machine components for overload
--	---

### Further options available



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

For more information, please refer to your machine manufacturer'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 manufacturer 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

#### 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, the terms of use agreed upon between HEIDENHAIN and Softing Industrial Automation GmbH shall additionally apply and prevail.

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 and modified cycle functions of software 81760x-16 " 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 downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

## New and modified cycle functions of software 81760x-16



### 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:

#### Modified functions:

- Within the **CONTOUR DEF** function, you can exclude certain areas **V** (void) from machining. These areas may be contours in cast parts or machining operations from previous steps, for example.  
**Further information:** "SL or OCM cycles with simple contour formula", Page 400
- In Cycle **12 PGM CALL** (ISO: G39), you use the **SYNTAX** soft key to place paths within quotation marks. To separate folders and files within paths, both the \ and the / character are permitted.  
**Further information:** "Cycle 12 PGM CALL ", Page 408
- The parameter **Q357 CLEARANCE TO SIDE** has been added to Cycle **202 BORING** (ISO: **G202**, option 19). This parameter allows you to define how far the control retracts the tool at the bottom of the hole in the working plane. This parameter is only effective if the parameter **Q214 DISENGAGING DIRECTN** has been defined.  
**Further information:** "Cycle 202 BORING (option 19)", Page 76
- The parameter **Q373 FEED AFTER REMOVAL** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**, option 19). This parameter is used to define the feed rate for returning to the advanced stop distance after chip removal.  
**Further information:** "Cycle 205 UNIVERSAL PECKING (option 19)", Page 89
- The parameter **Q370 TOOL PATH OVERLAP** has been added to Cycle **208 BORE MILLING** (ISO: **G208**, option 19). Use this parameter to define the lateral infeed.

- In Cycle **224 DATAMATRIX CODE PATTERN** (ISO: **G224**, option 9), you can output the following system data as variables:
  - Current date
  - Current time
  - Current calendar week number
  - Name and path of an NC program
  - Current count

**Further information:** "Outputting variable texts in DataMatrix codes", Page 254

- Cycle **225 ENGRAVING** (ISO: **G225**) has been enhanced:
  - The parameter **Q202 MAX. PLUNGING DEPTH** allows you to define the maximum plunging depth.
  - The parameter **Q367 TEXT POSITION** now includes the input options **7, 8** and **9**. With these values, you can set the reference of the engraving text onto the horizontal centerline.
  - The approach behavior was changed. If the tool is below the **2ND SET-UP CLEARANCE**, the control first positions to the 2nd set-up clearance **Q204** and then to the starting position in the working plane.

**Further information:** "Cycle 225 ENGRAVING ", Page 415

- If, in Cycle **233 FACE MILLING** (ISO: **G233**, option 19), the parameter **Q389** has been defined with the value **2** or **3** and a lateral limit has been defined in addition, then the control approaches the contour or departs from it on an arc with **Q207 FEED RATE MILLING**.

**Further information:** "Cycle 233 FACE MILLING (option 19)", Page 206

- If a measurement in Cycle **238 MEASURE MACHINE STATUS** (ISO: **G238**, option 155) was not performed correctly, for example at a feed rate override of 0 %, you can repeat the cycle.

**Further information:** "Cycle 238 MEASURE MACHINE STATUS (option 155)", Page 428

- Cycle **240 CENTERING** (ISO: **G240**, option 19) has been enhanced in order to consider pre-drilled diameters.

The following parameters have been added:

- **Q342 ROUGHING DIAMETER**
- **Q253 F PRE-POSITIONING:** If parameter **Q342** is defined, feed rate for approaching the deepened starting point

**Further information:** "Cycle 240 CENTERING (option 19)", Page 112

- The parameters **Q429 COOLANT ON** and **Q430 COOLANT OFF** in Cycle **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**, option 19) have been enhanced. You can define a path for a user macro.

**Further information:** "Cycle 241 SINGLE-LIP D.H.DRLNG (option 19)", Page 102

- An input value of 2 has been added to parameter **Q575 INFEED STRATEGY** in Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167). With this input option, the control calculates the machining sequence so that the tooth length of the tool is used to its full extent.

**Further information:** "Cycle 272 OCM ROUGHING (option 167) ", Page 315

- Certain cycles permit you to enter tolerances. In the following cycles, you can define dimensions, tolerances according to DIN EN ISO 286-2 or general tolerances according to DIN ISO 2768-1:
  - Cycle **208 BORE MILLING** (ISO: **G208**, option 19)
  - Cycle **1271 OCM RECTANGLE** (ISO: **G1271**, option 167)
  - Cycle **1272 OCM CIRCLE** (ISO: **G1272**, option 167)
  - Cycle **1273 OCM SLOT / RIDGE** (ISO: **G1273**, option 167)
  - Cycle **1278 OCM POLYGON** (ISO: **G1278**, option 167)

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

#### New functions

- Cycle **1400 POSITION PROBING** (ISO: **G1400**)  
This cycle allows you to probe a single position. You can transfer the acquired values to the active line of the preset table.
- Cycle **1401 CIRCLE PROBING** (ISO: **G1401**)  
This cycle allows you to determine the center point of a hole or a stud. You can transfer the acquired values to the active line of the preset table.
- Cycle **1402 SPHERE PROBING** (ISO: **G1402**)  
This cycle allows you to determine the center point of a sphere. You can transfer the acquired values to the active line of the preset table.
- Cycle **1412 INCLINED EDGE PROBING** (ISO: **G1412**)  
This cycle allows you to ascertain a workpiece misalignment by probing two points on an inclined edge.
- Cycle **1493 EXTRUSION PROBING** (ISO: **G1493**)  
This cycle allows you to define an extrusion. With an active extrusion, the control repeats the touch points along a direction over a defined length.

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



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

### Modified functions

- The unit of measurement of the main program can be seen in the header of the log file of the Probing Cycles **14xx** and **42x**
- If a basic rotation is active in the workpiece preset, the control shows an error message during execution of Cycles **451 MEASURE KINEMATICS** (ISO: **G451**, option 48), **452 PRESET COMPENSATION** (ISO: **G452**, option 48). The control resets the basic rotation to 0 during continuation of the program.
- The parameter **Q523 TT-POSITION** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define the position of the tool touch probe and, if desired, to transfer the position to the machine parameter **centerPos** after calibration.
- The Cycles **1420 PROBING IN PLANE** (ISO: **G1420**), **1410 PROBING ON EDGE** (ISO: **G1410**), **1411 PROBING TWO CIRCLES** (ISO: **G1411**) have been enhanced:
  - In these cycles, you can define tolerances according to DIN EN ISO 286-2 or general tolerances according to DIN ISO 2768-1.
  - If you defined the value 2 in parameter **Q1125 CLEAR. HEIGHT MODE**, the control positions the touch probe at rapid traverse **FMAX** from the touch probe table to the set-up clearance.

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



# 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!

- Test your program before executing it



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	68
	Cycles for tapping, thread cutting and thread milling	120
	Cycles for milling pockets, studs, slots, and face milling	162
	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	222
	SL (Subcontour List) cycles for machining contours that consist of multiple overlapping subcontours as well as cycles for cylinder surface machining and trochoidal milling	262
	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	242
	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, determining the load,	406











- 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	<b>Further information:</b> User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic workpiece presetting	<b>Further information:</b> User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic workpiece inspection	<b>Further information:</b> User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Special cycles	<b>Further information:</b> User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Touch probe calibration	<b>Further information:</b> User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic kinematics measurement	<b>Further information:</b> User's Manual for Programming of Measuring Cycles for Workpieces and Tools
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	<b>Further information:</b> 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 manufacturer	

# 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) at the same time, proceed as follows:

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.

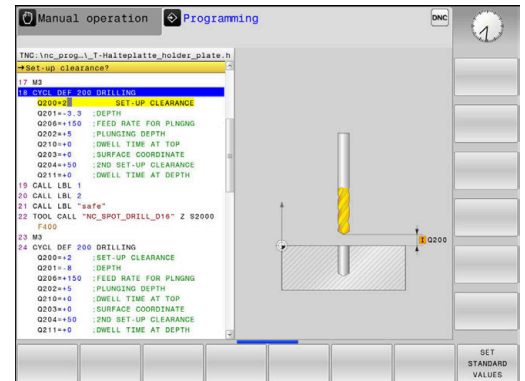
**Further information:** "Calling a cycle", Page 43



## 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.



### NOTICE

#### Danger of collision!

You can program variables as input values in HEIDENHAIN cycles. Using variables outside of the recommended input ranges can lead to collisions.

- ▶ Only use the input ranges recommended by HEIDENHAIN
- ▶ Pay attention to the HEIDENHAIN documentation
- ▶ Check the machining sequence using a simulation

## 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

11 CYCL DEF 200 DRILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;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	;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 requirements must be observed. They are detailed in the descriptions and overview tables 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:



- ▶ Press the **CYCL CALL** key



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

### Calling a cycle with CYCL CALL PAT

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

**Further information:** "Pattern definition with PATTERN DEF",  
Page 54

**Further information:** User's Manual for **Klartext Programming** or **ISO Programming**

### 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 **SELECT 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 **SELECT 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, **SELECT 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** function 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.

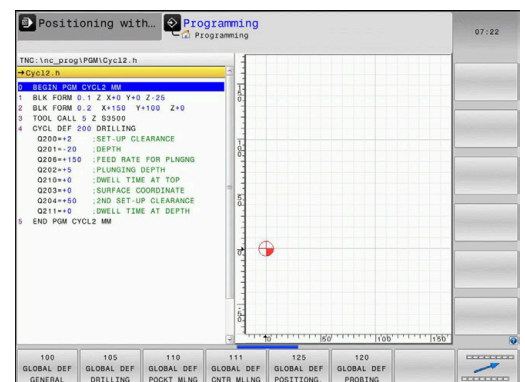
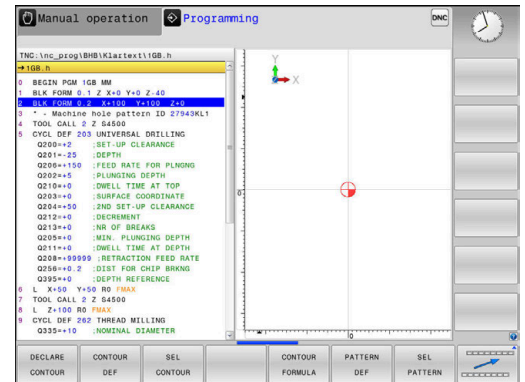
The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF GENERAL Definition of generally valid cycle parameters	49
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	50
110 GLOBAL DEF POCKET MILLING	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	51
111 GLOBAL DEF CONTR MILLING	GLOBAL DEF CONTOUR MILLING Definition of specific contour-milling cycle parameters	52
125 GLOBAL DEF POSITIONING.	GLOBAL DEF POSITIONING Definition of the positioning behavior with <b>CYCL CALL PAT</b>	52
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	53

### Entering GLOBAL DEF

Proceed as follows:

- ▶ Press the **Editor** 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

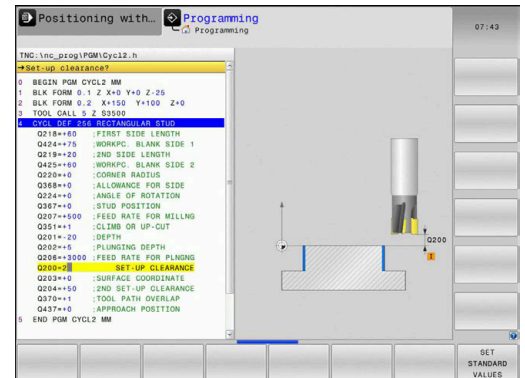


## Using GLOBAL DEF information

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 **2xx** machining cycles and touch probe cycles **451, 452**

Help graphic	Parameter
	<b>Q200 Set-up clearance?</b> Distance between tool tip and workpiece surface. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q204 2nd set-up clearance?</b> Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q253 Feed rate for pre-positioning?</b> Feed rate at which the control moves the tool within a cycle. Input: <b>0...99999.999</b> or <b>FMAX, FAUTO</b>
	<b>Q208 Feed rate for retraction?</b> Feed rate at which the control retracts the tool. Input: <b>0...99999.999</b> or <b>FMAX, FAUTO</b>

### Example

11 GLOBAL DEF 100 GENERAL ~	
Q200=+2	;SET-UP CLEARANCE ~
Q204=+50	;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**.

Help graphic	Parameter
	<b>Q256 Retract dist. for chip breaking?</b> Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: <b>0.1...99999.9999</b>
	<b>Q210 Dwell time at the top?</b> Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input: <b>0...3600.0000</b>
	<b>Q211 Dwell time at the depth?</b> Time in seconds that the tool remains at the hole bottom. Input: <b>0...3600.0000</b>

### 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**

Help graphic	Parameter
	<b>Q370 Path overlap factor?</b> <b>Q370</b> x tool radius = stepover factor k. Input: <b>0.1...1999</b>
	<b>Q351 Direction? Climb=+1, Up-cut=-1</b> Type of milling operation. The direction of spindle rotation is taken into account. <b>+1</b> = climb milling <b>-1</b> = up-cut milling (If you enter 0, climb milling is performed.) Input: <b>-1, 0, +1</b>
	<b>Q366 Plunging strategy (0/1/2)?</b> Type of plunging strategy: <b>0</b> : Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle <b>ANGLE</b> defined in the tool table. <b>1</b> : Helical plunging. In the tool table, the plunging angle <b>ANGLE</b> for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message <b>2</b> : Reciprocating plunge. In the tool table, the plunging angle <b>ANGLE</b> 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. Input: <b>0, 1, 2</b>

### Example

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

## Global data for milling operations with contour cycles

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

Help graphic	Parameter
	<b>Q2 Path overlap factor?</b> Q2 x tool radius = stepover factor k Input: <b>0.0001...1.9999</b>
	<b>Q6 Set-up clearance?</b> Distance between tool tip and the top surface of the workpiece. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q7 Clearance height?</b> Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q9 Direction of rotation? cw = -1</b> Machining direction for pockets <ul style="list-style-type: none"> <li>■ <b>Q9</b> = -1 up-cut milling for pocket and island</li> <li>■ <b>Q9</b> = +1 climb milling for pocket and island</li> </ul> Input: <b>-1, 0, +1</b>

### 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.

Help graphic	Parameter
	<b>Q345 Select positioning height (0/1)</b> Retraction in the tool axis at the end of a machining step, return to the 2nd set-up clearance or to the position at the beginning of the unit. Input: <b>0, 1</b>

### 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**

Help graphic	Parameter
	<b>Q320 Set-up clearance?</b> Additional distance between touch point and ball tip. <b>Q320</b> is in addition to the <b>SET_UP</b> column in the touch probe table. This value has an incremental effect. Input: <b>0...99999.9999</b> or <b>PREDEF</b>
	<b>Q260 Clearance height?</b> Coordinate in the tool axis at which no collision between tool and workpiece (fixtures) can occur. The value has an absolute effect. Input: <b>-99999.9999...+99999.9999</b> or <b>PREDEF</b>
	<b>Q301 Move to clearance height (0/1)?</b> Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: <b>0, 1</b>

### Example

<b>11 GLOBAL DEF 120 PROBING ~</b>	
<b>Q320=+0</b>	<b>;SET-UP CLEARANCE ~</b>
<b>Q260=+100</b>	<b>;CLEARANCE HEIGHT ~</b>
<b>Q301=+1</b>	<b>;MOVE TO CLEARANCE</b>

### 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 tool 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	56
	ROW Definition of a single row, straight or rotated	57
	PATTERN Definition of a single pattern, straight, rotated or distorted	58
	FRAME Definition of a single frame, straight, rotated or distorted	60
	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 43

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 tool axis position 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:

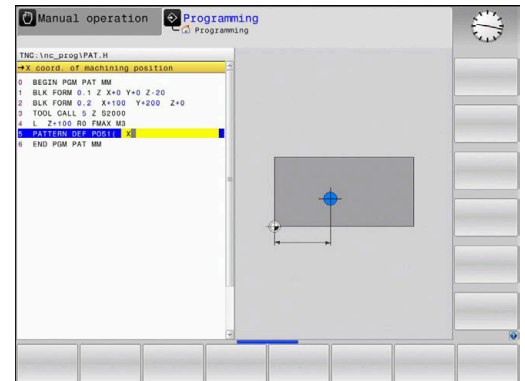
- You can use the mid-program startup function to select any point at which you want to start or continue machining.  
**Further information:** User's Manual for Setup, Testing and Running NC programs

## 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.



### Help graphic

### Parameter

POS1: **X coord. of machining position**

Enter the X coordinate as an absolute value.

Input: **-999999999...+999999999**

POS1: **Y coord. of machining position**

Enter the Y coordinate as an absolute value.

Input: **-999999999...+999999999**

POS1: **Coordinate of workpiece surface**

Enter the Z coordinate as an absolute value at which machining starts.

Input: **-999999999...+999999999**

POS2: **X coord. of machining position**

Enter the X coordinate as an incremental or absolute value.

Input: **-999999999...+999999999**

POS2: **Y coord. of machining position**

Enter the Y coordinate as an incremental or absolute value.

Input: **-999999999...+999999999**

POS2: **Coordinate of workpiece surface**

Enter the Z coordinate as an incremental or absolute value.

Input: **-999999999...+999999999**

### Example

11 PATTERN DEF ~

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

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

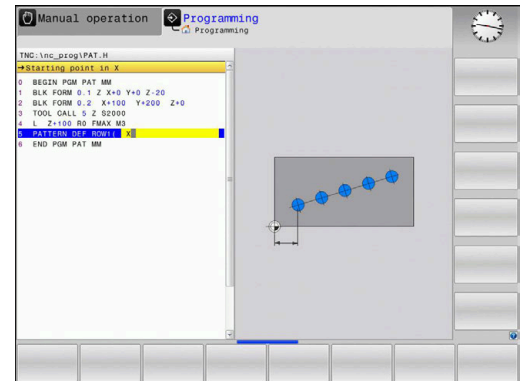


## Defining a single row



Programming and operating note:

- 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.



### Help graphic

### Parameter

#### Starting point in X

Coordinate of the starting point of the row in the X axis. The value has an absolute effect.

Input: **-99999.999999...+99999.999999**

#### Starting point in Y

Coordinate of the starting point of the row in the Y axis. The value has an absolute effect.

Input: **-99999.999999...+99999.999999**

#### Spacing of machining positions

Distance (incremental) between the machining positions. Enter a positive or negative value

Input: **-999999999...+999999999**

#### Number of operations

Total number of machining operations

Input: **0...999**

#### Rot. position of entire pattern

Angle of rotation around the entered starting point. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). Enter a positive or negative absolute value

Input: **-360.000...+360.000**

#### Coordinate of workpiece surface

Enter the Z coordinate as an absolute value at which machining starts

Input: **-999999999...+999999999**

### Example

```
11 PATTERN DEF ~
```

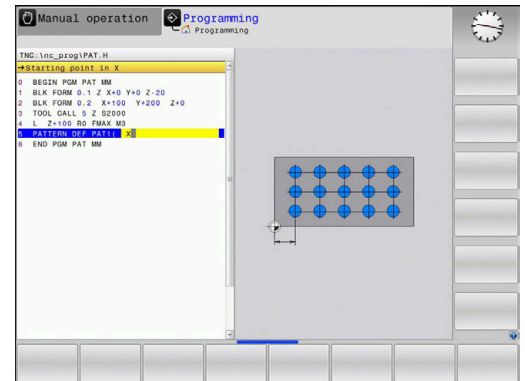
```
ROW1( X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0 )
```

## 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.



### Help graphic

### Parameter

#### Starting point in X

Absolute coordinate of the pattern starting point in the X axis

Input: **-999999999...+999999999**

#### Starting point in Y

Absolute coordinate of the pattern starting point in the Y axis

Input: **-999999999...+999999999**

#### Spacing of machining positions X

Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value

Input: **-999999999...+999999999**

#### Spacing of machining positions Y

Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value

Input: **-999999999...+999999999**

#### Number of columns

Total number of columns in the pattern

Input: **0...999**

#### Number of rows

Total number of rows in the pattern

Input: **0...999**

#### Rot. position of entire pattern

Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). Enter a positive or negative absolute value

Input: **-360.000...+360.000**

#### Rotary pos. ref. ax.

Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value

Input: **-360.000...+360.000**

**Help graphic****Parameter****Rotary pos. minor ax.**

Angle of rotation around 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

Input: **-360.000...+360.000**

**Coordinate of workpiece surface**

Enter the Z coordinate as an absolute value at which machining starts.

Input: **-999999999...+999999999**

**Example**

```
11 PATTERN DEF ~
```

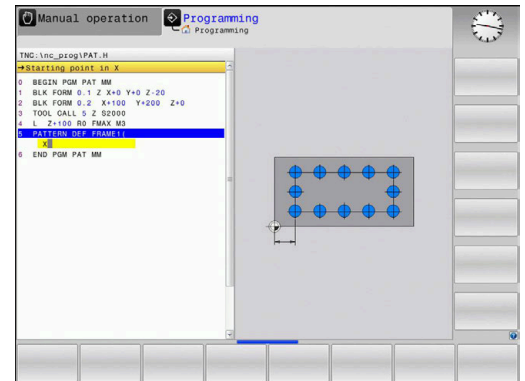
```
PAT1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0  
      ROTY+0 Z+0 )
```

## 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.



### Help graphic

### Parameter

#### Starting point in X

Absolute coordinate of the frame starting point in the X axis

Input: -999999999...+999999999

#### Starting point in Y

Absolute coordinate of the frame starting point in the Y axis

Input: -999999999...+999999999

#### Spacing of machining positions X

Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value

Input: -999999999...+999999999

#### Spacing of machining positions Y

Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value

Input: -999999999...+999999999

#### Number of columns

Total number of columns in the pattern

Input: 0...999

#### Number of rows

Total number of rows in the pattern

Input: 0...999

#### Rot. position of entire pattern

Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). Enter a positive or negative absolute value

Input: -360.000...+360.000

#### Rotary pos. ref. ax.

Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value.

Input: -360.000...+360.000

Help graphic	Parameter
	<p><b>Rotary pos. minor ax.</b></p> <p>Angle of rotation around 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.</p> <p>Input: <b>-360.000...+360.000</b></p>
	<p><b>Coordinate of workpiece surface</b></p> <p>Enter the Z coordinate as an absolute value at which machining starts</p> <p>Input: <b>-999999999...+999999999</b></p>

**Example**

```
11 PATTERN DEF ~
```

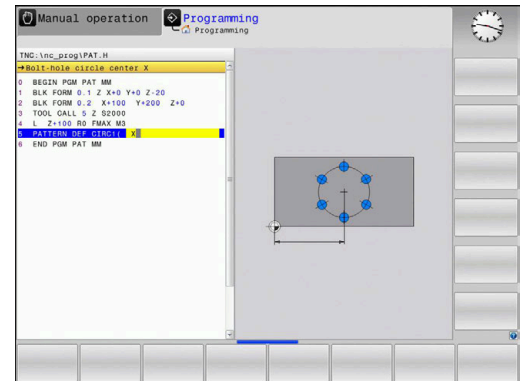
```
FRAME1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0
        ROTY+0 Z+0 )
```

## 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.



### Help graphic

### Parameter

#### Bolt-hole circle center X

Absolute coordinate of the circle center point in the X axis

Input: -999999999...+999999999

#### Bolt-hole circle center Y

Absolute coordinate of the circle center point in the Y axis

Input: -999999999...+999999999

#### Bolt-hole circle diameter

Diameter of the bolt hole circle

Input: 0...999999999

#### Starting angle

Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value

Input: -360.000...+360.000

#### Number of operations

Total number of machining positions on the circle

Input: 0...999

#### Coordinate of workpiece surface

Enter the Z coordinate as an absolute value at which machining starts.

Input: -999999999...+999999999

### Example

```
11 PATTERN DEF ~
```

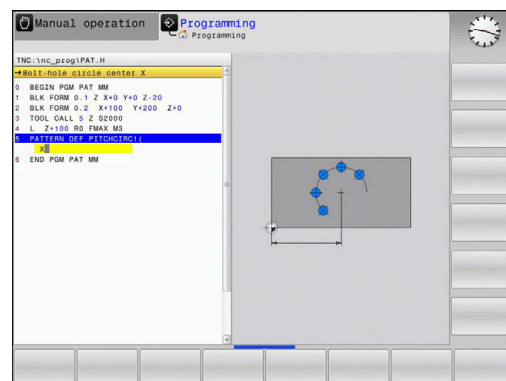
```
CIRC1( X+25 Y+33 D80 START+45 NUM8 Z+0 )
```

## 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.



### Help graphic

### Parameter

#### Bolt-hole circle center X

Absolute coordinate of the circle center point in the X axis

Input: **-999999999...+999999999**

#### Bolt-hole circle center Y

Absolute coordinate of the circle center point in the Y axis

Input: **-999999999...+999999999**

#### Bolt-hole circle diameter

Diameter of the bolt hole circle

Input: **0...999999999**

#### Starting angle

Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value

Input: **-360.000...+360.000**

#### 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 Stopping angle (switch via soft key)

Input: **-360.000...+360.000**

#### Number of operations

Total number of machining positions on the circle

Input: **0...999**

#### Coordinate of workpiece surface

Enter the Z coordinate at which machining starts.

Input: **-999999999...+999999999**

### Example

```
11 PATTERN DEF ~
```

```
PITCHCIRC1( X+25 Y+33 D80 START+45 STEP+30 NUM8 Z+0 )
```

## 3.4 Point tables with cycles

### Application with cycles

With a point table you can execute one or more cycles in sequence on an irregular point pattern.

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

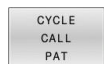
### Calling a cycle in connection with point tables

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
	Cycle 200 DRILLING <ul style="list-style-type: none"> <li>■ Basic hole</li> <li>■ Input of the dwell time at top and bottom</li> <li>■ Depth reference selectable</li> </ul>	70
	Cycle 201 REAMING (option 19) <ul style="list-style-type: none"> <li>■ Reaming a hole</li> <li>■ Input of the dwell time at bottom</li> </ul>	74
	Cycle 202 BORING (option 19) <ul style="list-style-type: none"> <li>■ Boring a hole</li> <li>■ Input of the retraction feed rate</li> <li>■ Input of the dwell time at bottom</li> <li>■ Input of the retracting movement</li> </ul>	76
	Cycle 203 UNIVERSAL DRILLING (option 19) <ul style="list-style-type: none"> <li>■ Degression – hole with decreasing infeed</li> <li>■ Input of the dwell time at top and bottom</li> <li>■ Input of chip breaking behavior</li> <li>■ Depth reference selectable</li> </ul>	80
	Cycle 204 BACK BORING (option 19) <ul style="list-style-type: none"> <li>■ Machining a counterbore on the underside of the workpiece</li> <li>■ Input of the dwell time</li> <li>■ Input of the retracting movement</li> </ul>	85
	Cycle 205 UNIVERSAL PECKING (option 19) <ul style="list-style-type: none"> <li>■ Degression – hole with decreasing infeed</li> <li>■ Input of chip breaking behavior</li> <li>■ Input of a deepened starting point</li> <li>■ Input of an advanced stop distance</li> </ul>	89

Soft key	Cycle	Page
	Cycle 208 BORE MILLING (option 19) <ul style="list-style-type: none"> <li>■ Milling of a hole</li> <li>■ Input of a pre-drill diameter</li> <li>■ Climb or up-cut milling selectable</li> </ul>	97
	Cycle 241 SINGLE-LIP D.H.DRLNG (option 19) <ul style="list-style-type: none"> <li>■ Drilling with single-lip deep hole drill</li> <li>■ Deepened starting point</li> <li>■ Direction of rotation and rotational speed for moving into and retracting from the hole</li> <li>■ Input of the dwell depth</li> </ul>	102
	Cycle 240 CENTERING (option 19) <ul style="list-style-type: none"> <li>■ Drilling a center hole</li> <li>■ Input of the centering diameter or depth</li> <li>■ Input of the dwell time at bottom</li> </ul>	112

## 4.2 Cycle 200 DRILLING

### ISO programming

#### G200

### Application

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

### Cycle sequence

- 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**

### Notes

#### 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 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.

**Notes on programming**

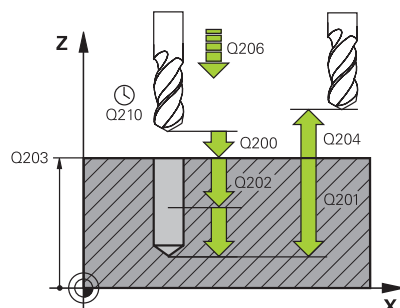
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- This cycle monitors the defined usable 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

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

#### Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

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

Input: **0...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.

Input: **0...3600.0000** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active preset. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000** or **PREDEF**



## Help graphic

## Parameter

**Q395 Diameter as reference (0/1)?**

Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the 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 table TOOL.T.

**0** = Depth referenced to tool tip

**1** = Depth referenced to the cylindrical part of the tool

Input: **0, 1**

## Example

11 CYCL DEF 200 DRILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;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	;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 Cycle 201 REAMING (option 19)

### ISO programming

G201

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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

### Cycle sequence

- 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**

### Notes

#### 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

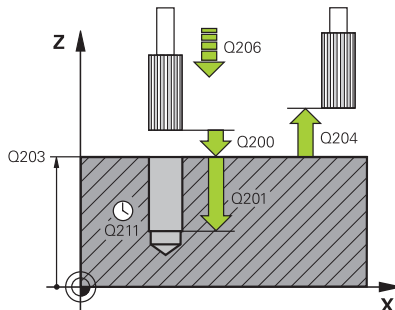
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

### Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while reaming

Input: **0...99999.999** or **FAUTO, FU**

#### Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000** or **PREDEF**

#### 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: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active preset. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

### Example

11 CYCL DEF 201 REAMING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;RETRACTION FEED RATE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE
12 L X+30 Y+20 FMAX M3	
13 CYCL CALL	

## 4.4 Cycle 202 BORING (option 19)

### ISO programming

#### G202

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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 a dwell time at the bottom of the hole.

### Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the safety clearance **Q200** above the workpiece **Q203 SURFACE COORDINATE**
- 2 The tool drills to the programmed depth at the feed rate for plunging **Q201**
- 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 **Q214 DISENGAGING DIRECTN** is defined, the control retracts in the programmed direction by the value in **CLEARANCE TO SIDE Q357**
- 6 Then the control moves the tool at the retraction feed rate **Q208** to the set-up clearance **Q200**
- 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

### Notes

#### 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** operating mode). 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 the tool will not be retracted!

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

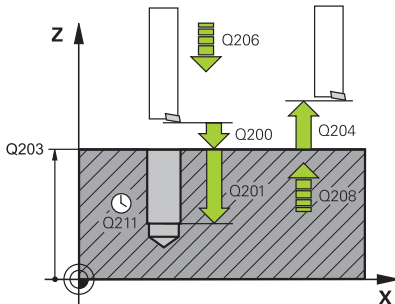
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- If the M7 or M8 function was active before calling the cycle, the 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.
- If **Q214 DISENGAGING DIRECTN** is not 0, **Q357 CLEARANCE TO SIDE** is in effect.

**Notes on programming**

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while boring

Input: **0...99999.999** or **FAUTO, FU**

#### Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000** or **PREDEF**

#### 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: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q214 Disengaging directn (0/1/2/3/4)?

Specify the direction in which the control retracts the tool at the hole bottom (after carrying out an oriented spindle stop)

**0:** Do not retract tool

**1:** Retract tool in negative main axis direction

**2:** Retract tool in negative secondary axis direction

**3:** Retract tool in positive main axis direction

**4:** Retract tool in positive secondary axis direction

Input: **0, 1, 2, 3, 4**

#### Q336 Angle for spindle orientation?

Angle to which the control positions the tool before retracting it. The value has an absolute effect.

Input: **0...360**

## Help graphic

## Parameter

**Q357 Safety clearance to the side?**

Distance between tool tooth and the wall. This value has an incremental effect.

Only in effect if **Q214 DISENGAGING DIRECTN** is not 0.

Input: **0...99999.9999**

## Example

11 L Z+100 R0 FMAX
12 CYCL DEF 202 BORING ~
Q200=+2 ;SET-UP CLEARANCE ~
Q201=-20 ;DEPTH ~
Q206=+150 ;FEED RATE FOR PLNGNG ~
Q211=+0 ;DWELL TIME AT DEPTH ~
Q208=+99999 ;RETRACTION FEED RATE ~
Q203=+0 ;SURFACE COORDINATE ~
Q204=+50 ;2ND SET-UP CLEARANCE ~
Q214=+0 ;DISENGAGING DIRECTN ~
Q336=+0 ;ANGLE OF SPINDLE ~
Q357+0.2 ;CLEARANCE TO SIDE
13 L X+30 Y+20 FMAX M3
14 CYCL CALL
15 L X+80 Y+50 FMAX M99

## 4.5 Cycle 203 UNIVERSAL DRILLING (option 19)

ISO programming  
G203

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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

### Cycle sequence

#### 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 the **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**

## Notes

### 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

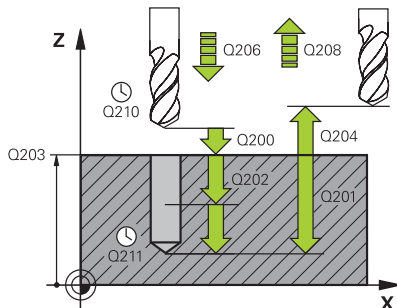
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

#### Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

## Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

### Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input:  $-99999.9999\dots + 99999.9999$

### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

### Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

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

Input: **0...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.

Input: **0...3600.0000** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

### Q212 Decrement?

Value by which the control decreases **Q202 Feed depth** after each infeed. This value has an incremental effect.

Input: **0...99999.9999**

**Q213 Nr of breaks before retracting?**

Number of chip breaks after which the control is to withdraw the tool from the hole for chip breaking. For chip breaking, the control retracts the tool each time by the value in **Q256**.

Input: **0...99999**

## Help graphic

## Parameter

**Q205 Minimum plunging depth?**

If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect.

Input: **0...99999.9999**

**Q211 Dwell time at the depth?**

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000** or **PREDEF**

**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: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

**Q256 Retract dist. for chip breaking?**

Value by which the control retracts the tool during chip breaking. This value has an incremental effect.

Input: **0...99999.999** or **PREDEF**

**Q395 Diameter as reference (0/1)?**

Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the 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 table TOOL.T.

**0** = Depth referenced to tool tip

**1** = Depth referenced to the cylindrical part of the tool

Input: **0, 1**

## 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 ~
Q210=+0	;DWELL TIME AT TOP ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q212=+0	;DECREMENT ~
Q213=+0	;NR OF BREAKS ~
Q205=+0	;MIN. PLUNGING DEPTH ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;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	

## 4.6 Cycle 204 BACK BORING (option 19)

### ISO programming

#### G204

### Application



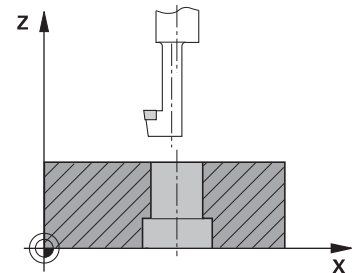
Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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 the 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**

## Notes

### 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** operating mode). 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.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- 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.



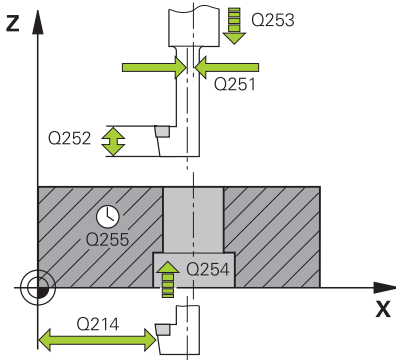
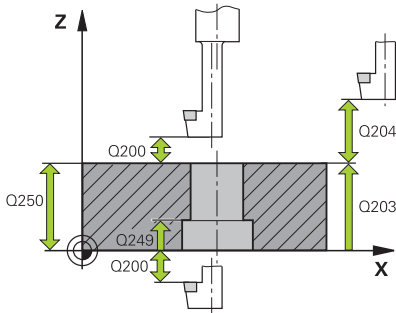
Enter the tool length measured up to the lower edge of the boring bar, not the cutting edge.

#### Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter depth determines the working direction. Note: If you enter a positive sign, the tool bores in the direction of the positive spindle axis.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q249 Depth of counterbore?

Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q250 Material thickness?

Height of the workpiece. Enter an incremental value.

Input: **0.0001...99999.9999**

#### Q251 Tool edge off-center distance?

Off-center distance of the boring bar. Refer to the tool data sheet. This value has an incremental effect.

Input: **0.0001...99999.9999**

#### Q252 Tool edge height?

Distance between underside of boring bar and main cutting tooth. Refer to the tool data sheet. This value has an incremental effect.

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q254 Feed rate for counterboring?

Traversing speed of the tool in mm/min during counterboring

Input: **0...99999.999** or **FAUTO, FU**

#### Q255 Dwell time in secs.?

Dwell time in seconds at the bottom of the bore hole

Input: **0...99999**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

## Help graphic

## Parameter

**Q214 Disengaging directn (0/1/2/3/4)?**

Specify the direction in which the control offsets the tool by the off-center distance (after orienting the spindle). Inputting 0 is not permitted

**1:** Retract tool in negative main axis direction

**2:** Retract tool in negative secondary axis direction

**3:** Retract tool in positive main axis direction

**4:** Retract tool in positive secondary axis direction

Input: **1, 2, 3, 4**

**Q336 Angle for spindle orientation?**

Angle at which the control positions the tool before it is plunged into or retracted from the bore hole. The value has an absolute effect.

Input: **0...360**

## 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=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q214=+0	;DISENGAGING DIRECTN ~
Q336=+0	;ANGLE OF SPINDLE
12 CYCL CALL	



## 4.7 Cycle 205 UNIVERSAL PECKING (option 19)

### ISO programming

G205

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

With this cycle, you can drill holes with decreasing infeed. The cycle may be executed with or without chip breaking. When the plunging depth is reached the cycle performs chip removal. If there is already a pilot hole then you can enter a deepened starting point. In this cycle, you can optionally define a dwell time at the bottom of the hole. This dwell time is used for chip breaking at the bottom of the hole.

**Further information:** "Chip removal and chip breaking", Page 95

### Cycle sequence

- 1 The control positions the tool in the tool axis at **FMAX** to the entered **Safety clearance Q200** above the **SURFACE COORDINATE Q203**.
- 2 If you program a deepened starting point in **Q379**, the control moves at the positioning feed rate **Q253 F PRE-POSITIONING** to the set-up clearance above the deepened starting point.
- 3 The tool drills at the programmed **Q206 FEED RATE FOR PLNGNG** to the plunging depth.
- 4 If you have programmed chip breaking, the control retracts the tool by the retraction value **Q256**.
- 5 Upon reaching the plunging depth, the control retracts the tool in the tool axis at the retraction feed rate **Q208** to the set-up clearance. The set-up clearance is above the **SURFACE COORDINATE Q203**.
- 6 The tool then moves at **Q373 FEED AFTER REMOVAL** to the entered advanced stop distance above the plunging depth last reached.
- 7 The tool drills at the feed in **Q206** to the next plunging depth. If a decrement Q212 is defined, the plunging depth is decreased after each infeed by the decrement.
- 8 The control repeats this procedure (steps 2 to 7) until the total drilling depth is reached.
- 9 If you entered a dwell time, the tool remains at the hole bottom for chip breaking. The control then retracts the tool at the retraction feed rate to the set-up clearance or the 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**.



After chip removal, the depth of the next chip breaking is referenced to the last plunging depth.

#### Example:

- **Q202 PLUNGING DEPTH** = 10 mm
- **Q257 DEPTH FOR CHIP BRKNG** = 4 mm

The control performs chip breaking at 4 mm and 8 mm. Chip removal is performed at 10 mm. Chip breaking is next performed at 14 mm and 18 mm, etc.

## Notes

### 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 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.



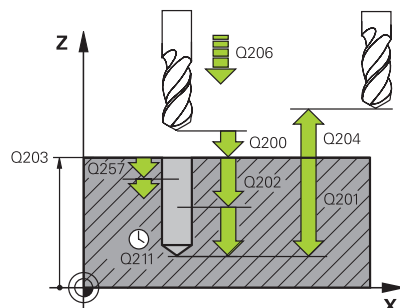
This cycle is not suitable for overlong drills. For overlong drills, use Cycle **241 SINGLE-LIP D.H.DRLNG**.

## Notes on programming

- 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 deepened 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.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth?

Distance between workpiece surface and bottom of hole (depends on parameter **Q395 DEPTH REFERENCE**). This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

#### Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

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

Input: **0...99999.9999**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q212 Decrement?

Value by which the control decreases the plunging depth **Q202**. This value has an incremental effect.

Input: **0...99999.9999**

#### Q205 Minimum plunging depth?

If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic	Parameter
	<b>Q258 Upper advanced stop distance?</b> Safety clearance above the last plunging depth to which the tool returns at <b>Q373 FEED AFTER REMOVAL</b> after first chip removal. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q259 Lower advanced stop distance?</b> Safety clearance above the last plunging depth to which the tool returns at <b>Q373 FEED AFTER REMOVAL</b> after last chip removal. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q257 Infeed depth for chip breaking?</b> Incremental depth at which the control performs chip breaking. This procedure is repeated until <b>DEPTH Q201</b> is reached. If <b>Q257</b> equals 0, the control will not perform chip breaking. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q256 Retract dist. for chip breaking?</b> Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: <b>0...99999.999</b> or <b>PREDEF</b>
	<b>Q211 Dwell time at the depth?</b> Time in seconds that the tool remains at the hole bottom. Input: <b>0...3600.0000</b> or <b>PREDEF</b>
	<b>Q379 Deepened starting point?</b> If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to <b>Q203 SURFACE COORDINATE</b> . The control moves at <b>Q253 F PRE-POSITIONING</b> to above the deepened starting point by the value <b>Q200 SET-UP CLEARANCE</b> . This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q253 Feed rate for pre-positioning?</b> Defines the tool traversing speed when positioning from <b>Q200 SET-UP CLEARANCE</b> to <b>Q379 STARTING POINT</b> (not equal to 0). Input in mm/min. Input: <b>0...99999.9999</b> or <b>FMAX, FAUTO, PREDEF</b>
	<b>Q208 Feed rate for retraction?</b> Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter <b>Q208 = 0</b> , the control retracts the tool at the feed rate specified in <b>Q206</b> . Input: <b>0...99999.9999</b> or <b>FMAX, FAUTO, PREDEF</b>

## Help graphic

## Parameter

**Q395 Diameter as reference (0/1)?**

Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the 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 table TOOL.T.

**0** = Depth referenced to tool tip

**1** = Depth referenced to the cylindrical part of the tool

Input: **0, 1**

**Q373 Post-chip-removal approach feed?**

Traversing speed of the tool when approaching the advanced stop distance after chip removal.

**0**: Move at **FMAX**

**>0**: Feed in mm/min

Input: **0...99999** or **FAUTO, FMAX, FU, FZ**

## Example

11 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;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	;DWELL TIME AT DEPTH ~
Q379=+0	;STARTING POINT ~
Q253=+750	;F PRE-POSITIONING ~
Q208=+99999	;RETRACTION FEED RATE ~
Q395=+0	;DEPTH REFERENCE ~
Q373=+0	;FEED AFTER REMOVAL

## Chip removal and chip breaking

### Chip removal

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

When the value entered in cycle parameter **Q202** is reached, the control performs chip removal. This means that the control always moves the tool to the retraction height, irrespective of the deepened starting point **Q379**. This height is calculated 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	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
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 ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool, end program
9 M30	
10 END PGM 205 MM	

### Chip breaking

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

When the value entered in cycle parameter **Q257** is reached, the control performs 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 **DEPTH Q201** is reached.

#### Example:

0 BEGIN PGM 205 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
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 ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool, end program
9 M30	
10 END PGM 205 MM	



## 4.8 Cycle 208 BORE MILLING (option 19)

### ISO programming

#### G208

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

With this cycle, you can mill holes. In this cycle, you can define an optional, pre-drilled diameter. You can also program tolerances for the nominal diameter.

### Cycle sequence

- 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 control moves on a semicircle for the first helical path while considering the path overlap **Q370**. The semicircle begins at the center of the hole.
- 3 The tool mills in a helix 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 the set-up clearance or to the 2nd set-up 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**



If you program **Q370=0** for the path overlap, the control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly.

### Tolerances

The control allows you to store tolerances in the parameter **Q335 NOMINAL DIAMETER**.

You can define the following tolerances:

Tolerance	Example	Manufacturing dimension
Deviations	10+0.01-0.015	9.9975
DIN EN ISO 286-2	10H7	10.0075
ISO 2768-1	10m	10.0000

Proceed as follows:

- ▶ Start cycle definition
- ▶ Define cycle parameters
- ▶ Press the **ENTER TEXT** soft key
- ▶ Enter a nominal dimension including tolerance



- Machining is performed at mid-tolerance.
- If you program an incorrect tolerance, the control interrupts machining with an error message.
- Pay attention to capitalization when entering tolerances.

## Notes

### 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.

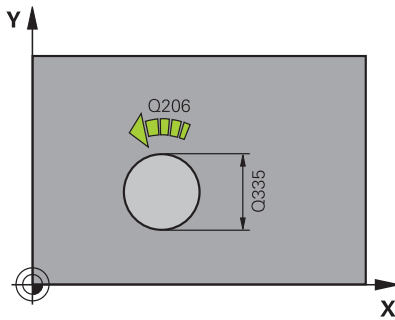
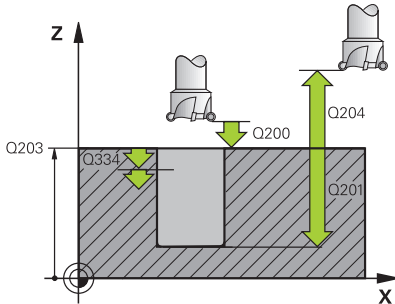
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have entered the bore hole diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation.
- An active mirror function **does not** influence the 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.

## Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between lower edge of tool and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min during helical drilling

Input: **0...99999.999** or **FAUTO, FU, FZ**

#### Q334 Feed per revolution of helix

Depth of the tool plunge with each helix ( $=360^\circ$ ). This value has an incremental effect.

Input: **0...99999.9999**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q335 Nominal diameter?

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. The value has an absolute effect. You can program a tolerance if needed.

**Further information:** "Tolerances", Page 98

Input: **0...99999.9999**

#### Q342 Roughing diameter?

Enter the dimension of the pre-drilled diameter. The value has an absolute effect.

Input: **0...99999.9999**

## Help graphic

## Parameter

**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)

Input: **-1, 0, +1** or **PREDEF**

**Q370 Path overlap factor?**

The control uses the path overlap factor to determine the stepover factor k.

**0**: The control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly.

**>0**: The control multiplies the factor by the active tool radius. The result is the stepover factor k.

Input: **0.1...1999** or **PREDEF**

## Example

11 CYCL DEF 208 BORE MILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q334=+0.25	;PLUNGING DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q335=+5	;NOMINAL DIAMETER ~
Q342=+0	;ROUGHING DIAMETER ~
Q351=+1	;CLIMB OR UP-CUT ~
Q370=+0	;TOOL PATH OVERLAP
12 CYCL CALL	

## 4.9 Cycle 241 SINGLE-LIP D.H.DRLNG (option 19)

ISO programming  
G241

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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 deepened starting point. You can define the direction of rotation and the rotational speed for moving into and retracting from the hole.

### Cycle sequence

- 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 108, 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 108
- 9 If programmed, the tool moves to 2nd set-up clearance at **FMAX**

## Notes

### 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

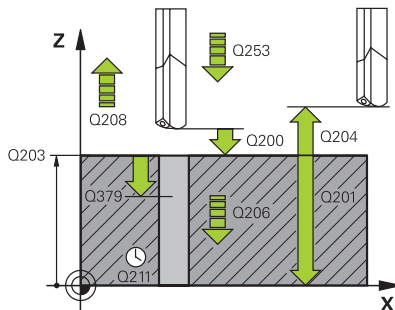
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

#### Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and **Q203 SURFACE COORDINATE**. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth?

Distance between **Q203 SURFACE COORDINATE** and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

#### Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active preset. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q379 Deepened starting point?

If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect.

Input: **0...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: **0...99999.9999** or **FMAX, FAUTO, PREDEF**



## Help graphic

## Parameter

**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: **0...99999.999** or **FMAX, FAUTO, PREDEF**

**Q426 Rot. dir. of entry/exit (3/4/5)?**

Rotational speed at which the tool is to rotate when moving into and retracting from the hole.

**3:** Spindle rotation with M3

**4:** Spindle rotation with M4

**5:** Movement with stationary spindle

Input: **3, 4, 5**

**Q427 Spindle speed of entry/exit?**

Rotational speed at which the tool is to rotate when moving into and retracting from the hole.

Input: **0...99999**

**Q428 Spindle speed for drilling?**

Desired speed for drilling.

Input: **0...99999**

**Q429 M function for coolant on?**

**>=0:** Miscellaneous function M for switching on the coolant. The control switches the coolant on when the tool has reached the set-up clearance **Q200** above the starting point **Q379**.

**"...":** Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically.

**Further information:** "User macro", Page 107

Input: **0...999**

**Q430 M function for coolant off?**

**>=0:** Miscellaneous function M for switching off the coolant. The control switches the coolant off if the tool is at the **DEPTH Q201**.

**"...":** Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically.

**Further information:** "User macro", Page 107

Input: **0...999**

## Help graphic

## Parameter

**Q435 Dwell depth?**

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**. This value has an incremental effect.

Input: **0...99999.9999**

**Q401 Feed rate factor in %?**

Factor by which the control reduces the feed rate after reaching **Q435 DWELL DEPTH**.

Input: **0.0001...100**

**Q202 Maximum plunging depth?**

Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect.

Input: **0...99999.9999**

**Q212 Decrement?**

Value by which the control decreases **Q202 Feed depth** after each infeed. This value has an incremental effect.

Input: **0...99999.9999**

**Q205 Minimum plunging depth?**

If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect.

Input: **0...99999.9999**

**Example**

11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q379=+0	;STARTING POINT ~
Q253=+750	;F PRE-POSITIONING ~
Q208=+1000	;RETRACTION FEED RATE ~
Q426=+5	;DIR. OF SPINDLE ROT. ~
Q427=+50	;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=+99999	;MAX. PLUNGING DEPTH ~
Q212=+0	;DECREMENT ~
Q205=+0	;MIN. PLUNGING DEPTH
12 CYCL CALL	

**User macro**

The user macro is another NC program.

A user macro contains a sequence of multiple instructions. With a macro, you can define multiple NC functions that the control executes. As a user, you create macros as an NC program.

Macros work in the same manner as NC programs that are called with the **PGM CALL** function, for example. You define a macro as an NC program with the file type \*.h or \*.i.

- HEIDENHAIN recommends using QL parameters in the macro. QL parameters have only a local effect for an NC program. If you use other types of variables in the macro, then changes may also have an effect on the calling NC program. In order to explicitly cause changes in the calling NC program, use Q or QS parameters with the numbers 1200 to 1399.
- Within the macro, you can read the value of the cycle parameters.

**Further information:** User's Manual for Conversational Programming

**Example of a user macro for coolant**

0 BEGIN PGM KM MM	
1 FN 18: SYSREAD QL100 = ID20 NR8	; Read the coolant level
2 FN 9: IF +QL100 EQU +1 GOTO LBL "Start"	; Query the coolant level; if coolant is active, jump to the <b>Start</b> LBL
3 M8	; Switch coolant on
7 CYCL DEF 9.0 DWELL TIME	
8 CYCL DEF 9.1 V.ZEIT3	
9 LBL "Start"	
10 END PGM RET MM	

**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 deepened starting point **Q379**. This value can be calculated as follows:  $0.2 \times Q379$ ; 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 starting point of drilling is calculated as follows:  $0.2 \times Q379 = 0.2 \times 2 = 0.4$ ; the starting point is 0.4 mm or inches above the deepened starting point. So if the deepened starting point is at -2, the control starts the drilling process at -1.6.

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	$0.2 \cdot 2 = 0.4$	-1.6
2	5	0	2	$0.2 \cdot 5 = 1$	-4
2	10	0	2	$0.2 \cdot 10 = 2$	-8
2	25	0	2	$0.2 \cdot 25 = 5$ ( <b>Q200</b> =2, $5 > 2$ , so the value 2 is used.)	-23
2	100	0	2	$0.2 \cdot 100 = 20$ ( <b>Q200</b> =2, $20 > 2$ , so the value 2 is used.)	-98
5	2	0	5	$0.2 \cdot 2 = 0.4$	-1.6
5	5	0	5	$0.2 \cdot 5 = 1$	-4
5	10	0	5	$0.2 \cdot 10 = 2$	-8
5	25	0	5	$0.2 \cdot 25 = 5$	-20
5	100	0	5	$0.2 \cdot 100 = 20$ ( <b>Q200</b> =5, $20 > 5$ , so the value 5 is used.)	-95
20	2	0	20	$0.2 \cdot 2 = 0.4$	-1.6
20	5	0	20	$0.2 \cdot 5 = 1$	-4
20	10	0	20	$0.2 \cdot 10 = 2$	-8
20	25	0	20	$0.2 \cdot 25 = 5$	-20
20	100	0	20	$0.2 \cdot 100 = 20$	-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 deepened 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 or inches above the deepened start point. So if the deepened 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 deepened 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	$0.8 \cdot 10 = 8$ ( <b>Q200</b> =2, $8 > 2$ , so the value 2 is used.)	-8
2	25	0	2	$0.8 \cdot 25 = 20$ ( <b>Q200</b> =2, $20 > 2$ , so the value 2 is used.)	-23
2	100	0	2	$0.8 \cdot 100 = 80$ ( <b>Q200</b> =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	$0.8 \cdot 10 = 8$ ( <b>Q200</b> =5, $8 > 5$ , so the value 5 is used.)	-5
5	25	0	5	$0.8 \cdot 25 = 20$ ( <b>Q200</b> =5, $20 > 5$ , so the value 5 is used.)	-20
5	100	0	5	$0.8 \cdot 100 = 80$ ( <b>Q200</b> =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	$0.8 \cdot 100 = 80$ ( <b>Q200</b> =20, $80 > 20$ , so the value 20 is used.)	-80

## 4.10 Cycle 240 CENTERING (option 19)

### ISO programming

#### G240

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Use Cycle **240 CENTERING** to machine center holes. You can specify the centering diameter or depth and an optional dwell time at the bottom. This dwell time is used for chip breaking at the bottom of the hole. If there is already a pilot hole then you can enter a deepened starting point.

### Cycle sequence

- 1 From the current position, the control positions the tool at rapid traverse **FMAX** in the working plane to the starting position.
- 2 The control positions the tool at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** above the workpiece surface **Q203**.
- 3 If you define **Q342 ROUGHING DIAMETER** not equal to 0, the control uses this value and the point angle of the tool **T-ANGLE** to calculate a deepened starting point. The control positions the tool at the **F PRE-POSITIONING Q253** feed rate to the deepened starting point.
- 4 The tool is centered at the programmed feed rate for plunging **F** to the programmed centering diameter or centering depth.
- 5 If a dwell time **Q211** is defined, the tool remains at the centering depth.
- 6 Finally, the tool is retracted to the set-up clearance or to the 2nd set-up 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**.

### Notes

#### 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 monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

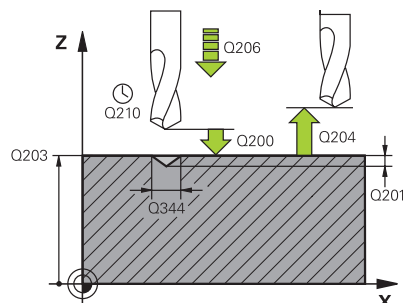


**Notes on programming**

- 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.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### 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

Input: **0, 1**

#### Q201 Depth?

Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if **Q343=0** is defined. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q344 Diameter of counterbore

Centering diameter. Only effective if **Q343=1** is defined.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while centering

Input: **0...99999.999** or **FAUTO, FU**

#### Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q342 Roughing diameter?

**0:** There is no hole

**>0:** Diameter of the pre-drilled hole

Input: **0...99999.9999**

## Help graphic

## Parameter

**Q253 Feed rate for pre-positioning?**

Traversing speed of the tool when approaching the deepened starting point. The speed is in mm/min.

Only in effect if **Q342 ROUGHING DIAMETER** is not 0.

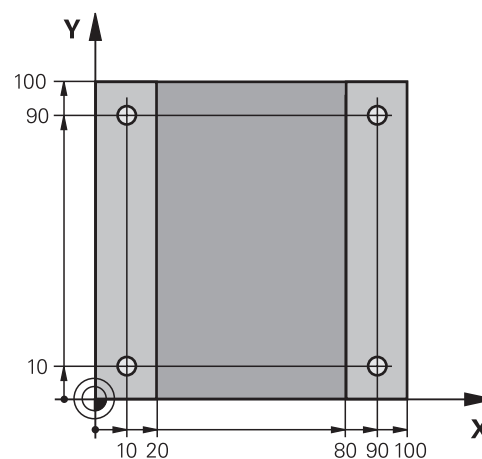
Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

## Example

11 CYCL DEF 240 CENTERING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q343=+1	;SELECT DIA./DEPTH ~
Q201=-2	;DEPTH ~
Q344=-10	;DIAMETER ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q342=+12	;ROUGHING DIAMETER ~
Q253=+500	;F PRE-POSITIONING
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 COORDINATEV	
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 cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the PATTERN DEF POS pattern definition. The control calls the drill hole coordinates 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)
- **GLOBAL DEF 125 POSITIONING:** 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.
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)

**Further information:** "Cycles: Tapping / Thread Milling", Page 119

0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+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 ~	
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 ~	
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 ~	
Q342=+0 ;ROUGHING DIAMETER ~	
Q253=+750 ;F PRE-POSITIONING	
7 GLOBAL DEF 125 POSITIONING ~	
Q345=+1 ;SELECT POS. HEIGHT	
8 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
9 L Z+100 R0 FMAX	; Retract the tool
10 TOOL CALL 227 Z S5000	; Tool call: drill (radius 2.4)

11 L X+50 R0 F5000	; Move tool to clearance height
12 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
13 CYCL CALL PAT F500 M3	; Cycle call in connection with the point pattern
14 L Z+100 R0 FMAX	; Retract the tool
15 TOOL CALL 263 Z S200	; Tool call: tap (radius 3)
16 L Z+100 R0 FMAX	; Move tool to clearance height
17 CYCL DEF 206 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	
18 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
19 L Z+100 R0 FMAX	; Retract the tool, end program
20 M30	
21 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
	Cycle 206 TAPPING <ul style="list-style-type: none"> <li>■ With a floating tap holder</li> <li>■ Input of the dwell time at bottom</li> </ul>	121
	Cycle 207 RIGID TAPPING <ul style="list-style-type: none"> <li>■ Without a floating tap holder</li> <li>■ Input of the dwell time at bottom</li> </ul>	124
	Cycle 209 TAPPING W/ CHIP BRKG (option 19) <ul style="list-style-type: none"> <li>■ Without a floating tap holder</li> <li>■ Input of chip breaking behavior</li> </ul>	128
	Cycle 262 THREAD MILLING (option 19) <ul style="list-style-type: none"> <li>■ Milling a thread into pre-drilled material</li> </ul>	135
	Cycle 263 THREAD MILLING/CNTRNG (option 19) <ul style="list-style-type: none"> <li>■ Milling a thread into pre-drilled material</li> <li>■ Machining a countersunk chamfer</li> </ul>	139
	Cycle 264 THREAD DRILLING/MILLING (option 19) <ul style="list-style-type: none"> <li>■ Drilling into solid material</li> <li>■ Milling a thread</li> </ul>	144
	Cycle 265 HEL. THREAD DRILLING/MILLING (option 19) <ul style="list-style-type: none"> <li>■ Milling a thread into solid material</li> </ul>	149
	Cycle 267 OUTSIDE THREAD MILLING (option 19) <ul style="list-style-type: none"> <li>■ Milling an external thread</li> <li>■ Machining a countersunk chamfer</li> </ul>	153



## 5.2 Cycle 206 TAPPING

### ISO programming

#### G206

### Application

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

### Cycle sequence

- 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.



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

### Notes

#### 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.
- 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.

**Notes on programming**

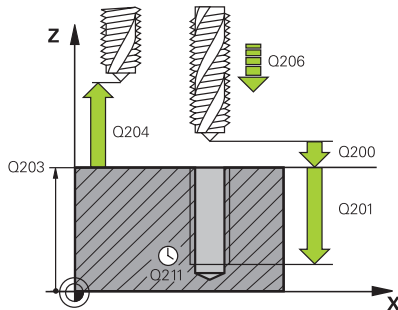
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

**Note regarding machine parameters**

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define 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)
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Guide value: 4 times the thread pitch

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool during tapping

Input: **0...99999.999** or **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: **0...3600.0000** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

### Example

11 CYCL DEF 206 TAPPING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-18	;DEPTH OF THREAD ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE
12 CYCL CALL	

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 Cycle 207 RIGID TAPPING

### ISO programming

#### G207

### Application



Refer to your machine manual.

Machine and control must be specially prepared by the machine manufacturer 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 sequence

- 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



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.

### Notes

#### 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 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.

#### Notes on programming

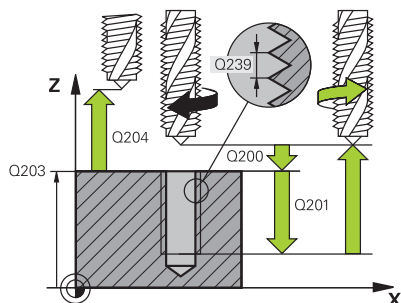
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

#### Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (spindle speed override is not active); the control then adjusts the spindle speed as required
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
  - **limitSpindleSpeed** (no. 113604): Spindle speed limit  
**True:** At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time.  
**False:** Limiting not active

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+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: **-99.9999...+99.9999**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

### Example

11 CYCL DEF 207 RIGID TAPPING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=- 18	;DEPTH OF THREAD ~
Q239=+1	;THREAD PITCH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE
12 CYCL CALL	

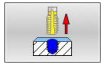
## Retracting after a program interruption

### 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.



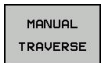
- ▶ Press **NC start**
- ▶ The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

### 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 Cycle 209 TAPPING W/ CHIP BRKG (option 19)

ISO programming  
G209

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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.

### Cycle sequence

- 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



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

### Notes

#### 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 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.

#### Notes on programming

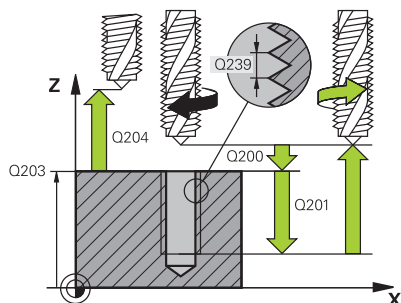
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.
- If you defined a speed factor for fast retraction in cycle parameter **Q403**, the control limits the speed to the maximum speed of the active gear stage.

#### Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define 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)
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

## Cycle parameters

### Help graphic



### Parameter

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+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: **-99.9999...+99.9999**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q257 Infeed depth for chip breaking?

Incremental depth at which the control performs chip breaking. This procedure is repeated until **DEPTH Q201** is reached. If **Q257** equals 0, the control will not perform chip breaking. This value has an incremental effect.

Input: **0...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 during chip breaking. If you enter **Q256 = 0**, the control retracts the tool completely from the hole (to set-up clearance) for chip breaking.

Input: **0...99999.9999**

#### Q336 Angle for spindle orientation?

Angle to which the control positions the tool before machining the thread. This allows you to re-cut the thread, if required. The value has an absolute effect.

Input: **0...360**

## Help graphic

## Parameter

**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 drill hole. Maximum increase to maximum speed of the active gear stage.

Input: **0.0001...10**

## Example

11 CYCL DEF 209 TAPPING W/ CHIP BRKG ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-18	;DEPTH OF THREAD ~
Q239=+1	;THREAD PITCH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q257=+0	;DEPTH FOR CHIP BRKNG ~
Q256=+1	;DIST FOR CHIP BRKNG ~
Q336=+0	;ANGLE OF SPINDLE ~
Q403=+1	;RPM FACTOR
12 CYCL CALL	

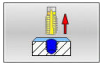
## Retracting after a program interruption

### 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.



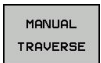
- ▶ Press **NC start**
- ▶ The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

### 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

### Requirements

- 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).
- If you are using a left-cutting tool (**M4**), the type of milling in **Q351** is reversed
- 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



Programming and operating notes:

- The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle **8 MIRRORING** in only one axis.
- 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.

## 5.6 Cycle 262 THREAD MILLING (option 19)

### ISO programming

#### G262

### Application



This function must be enabled and adapted by the machine manufacturer.

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

### Cycle sequence

- 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.

## Notes

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

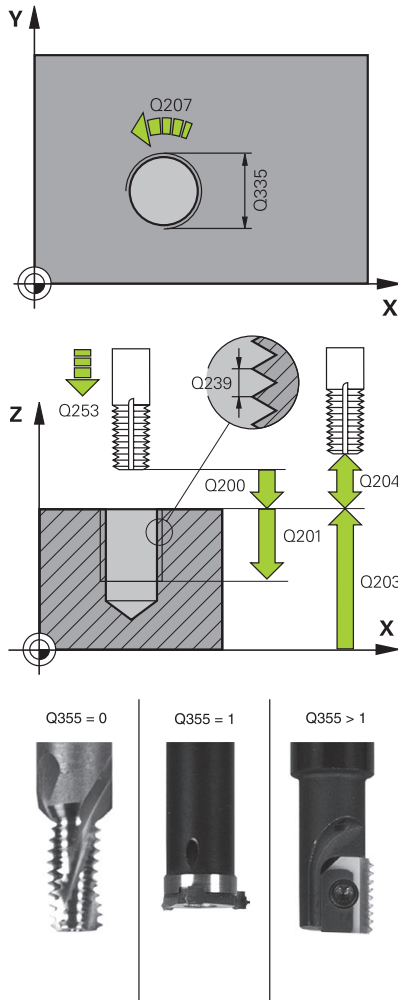
## Notes on programming

- 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.



## Cycle parameters

### Help graphic



### Parameter

#### Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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: **-99.9999...+99.9999**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q355 Number of threads per step?

Number of thread revolutions by which the tool is moved:

**0** = one helical line to the thread depth

**1** = continuous helical path over the entire length of the thread

**>1** = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch.

Input: **0...99999**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### 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)

Input: **-1, 0, +1** or **PREDEF**

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

## Help graphic

## Parameter

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min while milling

Input: **0...99999.999** or **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: **0...99999.999** or **FAUTO**

## Example

11 CYCL DEF 262 THREAD MILLING ~	
Q335=+5	;NOMINAL DIAMETER ~
Q239=+1	;THREAD PITCH ~
Q201=-18	;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=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q207=+500	;FEED RATE MILLING ~
Q512=+0	;FEED FOR APPROACH
12 CYCL CALL	

## 5.7 Cycle 263 THREAD MLLNG/CNTSNKG (option 19)

ISO programming  
G263

### Application



This function must be enabled and adapted by the machine manufacturer.

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

### Cycle sequence

- 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

## Notes

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

#### Notes on programming

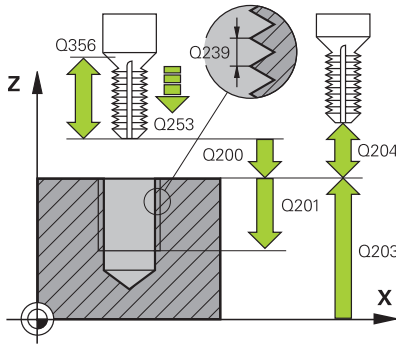
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.
- If you want to countersink at front, define the countersinking depth as 0.



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

## Cycle parameters

### Help graphic



### Parameter

#### Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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: **-99.9999...+99.9999**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q356 Countersinking depth?

Distance between tool point and the top surface of the workpiece. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### 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)

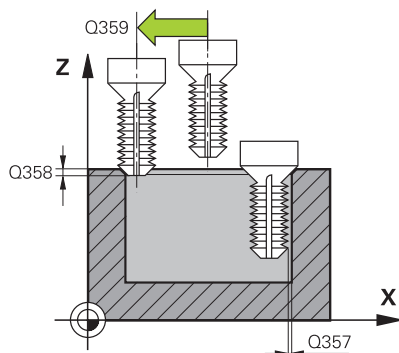
Input: **-1, 0, +1** or **PREDEF**

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

## Help graphic



## Parameter

**Q357 Safety clearance to the side?**

Distance between tool tooth and the wall. This value has an incremental effect.

Input: **0...99999.9999**

**Q358 Sinking depth at front?**

Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q359 Countersinking offset at front?**

Distance by which the control moves the tool center away from the center. This value has an incremental effect.

Input: **0...99999.9999**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q254 Feed rate for counterboring?**

Traversing speed of the tool in mm/min during counterboring

Input: **0...99999.999** or **FAUTO, FU**

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min while milling

Input: **0...99999.999** or **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: **0...99999.999** or **FAUTO**

**Example**

11 CYCL DEF 263 THREAD MLLNG/CNTSNKG ~	
Q335=+5	;NOMINAL DIAMETER ~
Q239=+1	;THREAD PITCH ~
Q201=-18	;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=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q254=+200	;F COUNTERBORING ~
Q207=+500	;FEED RATE MILLING ~
Q512=+0	;FEED FOR APPROACH
12 CYCL CALL	

## 5.8 Cycle 264 THREAD DRILLNG/MLLNG (option 19)

### ISO programming

G264

### Application



This function must be enabled and adapted by the machine manufacturer.

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

### Cycle sequence

- 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



## Notes

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

#### Notes on programming

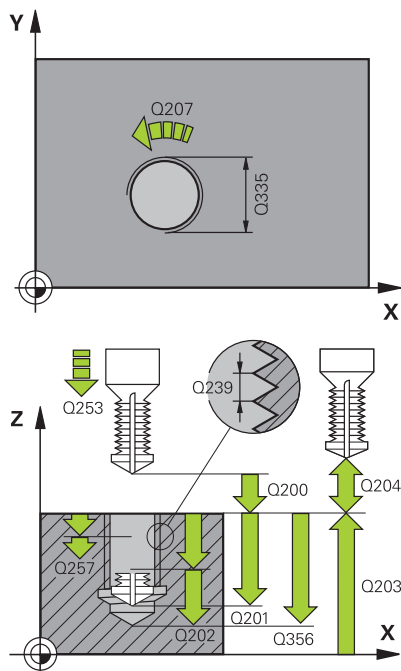
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.



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

## Cycle parameters

### Help graphic



### Parameter

#### Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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: **-99.9999...+99.9999**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q356 Total hole depth?

Distance between workpiece surface and hole bottom. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### 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)

Input: **-1, 0, +1** or **PREDEF**

#### Q202 Maximum plunging depth?

Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect.

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

Input: **0...99999.9999**

#### Q258 Upper advanced stop distance?

Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after first chip removal. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic	Parameter
	<b>Q257 Infeed depth for chip breaking?</b> Incremental depth at which the control performs chip breaking. This procedure is repeated until <b>DEPTH Q201</b> is reached. If <b>Q257</b> equals 0, the control will not perform chip breaking. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q256 Retract dist. for chip breaking?</b> Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: <b>0...99999.999</b> or <b>PREDEF</b>
	<b>Q358 Sinking depth at front?</b> Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q359 Countersinking offset at front?</b> Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q200 Set-up clearance?</b> Distance between tool tip and workpiece surface. This value has an incremental effect. Input: <b>0...99999.9999</b> or <b>PREDEF</b>
	<b>Q203 Workpiece surface coordinate?</b> Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q204 2nd set-up clearance?</b> Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: <b>0...99999.9999</b> or <b>PREDEF</b>
	<b>Q206 Feed rate for plunging?</b> Tool traversing speed in mm/min during plunging Input: <b>0...99999.999</b> or <b>FAUTO, FU</b>
	<b>Q207 Feed rate for milling?</b> Traversing speed of the tool in mm/min while milling Input: <b>0...99999.999</b> or <b>FAUTO</b>
	<b>Q512 Feed rate for approaching?</b> 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: <b>0...99999.999</b> or <b>FAUTO</b>

**Example**

11 CYCL DEF 264 THREAD DRILLNG/MLLNG ~	
Q335=+5	;NOMINAL DIAMETER ~
Q239=+1	;THREAD PITCH ~
Q201=-18	;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=+0	;DEPTH FOR CHIP BRKNG ~
Q256=+0.2	;DIST FOR CHIP BRKNG ~
Q358=+0	;DEPTH AT FRONT ~
Q359=+0	;OFFSET AT FRONT ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q207=+500	;FEED RATE MILLING ~
Q512=+0	;FEED FOR APPROACH
12 CYCL CALL	

## 5.9 Cycle 265 HEL. THREAD DRLG/MLG (option 19)

ISO programming  
G265

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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

## Notes

### 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

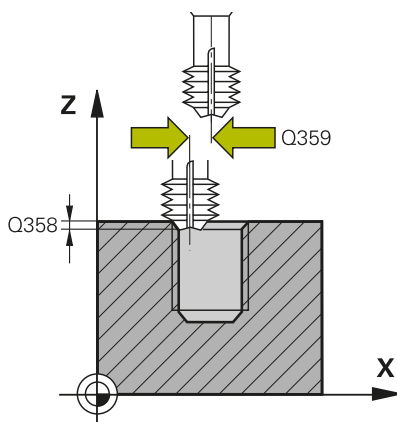
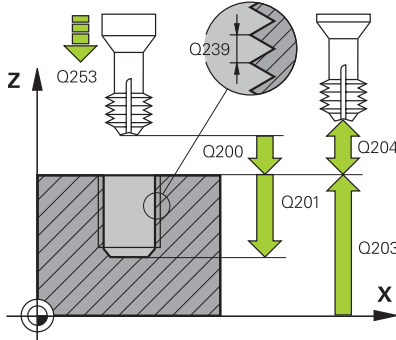
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.
- The type of milling (up-cut or climb) is determined by the thread (right-hand or left-hand thread) and the direction of tool rotation, since it is only possible to work in the direction of the tool.
- 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 Depth of thread
  - 2 Depth at front

#### Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

## Cycle parameters

### Help graphic



### Parameter

#### Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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: **-99.9999...+99.9999**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q358 Sinking depth at front?

Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q359 Countersinking offset at front?

Distance by which the control moves the tool center away from the center. This value has an incremental effect.

Input: **0...99999.9999**

#### Q360 Countersink (before/after:0/1)?

Execution of the chamfer

**0** = before thread machining

**1** = after thread machining

Input: **0, 1**

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

## Help graphic

## Parameter

**Q254 Feed rate for counterboring?**

Traversing speed of the tool in mm/min during counterboring

Input: **0...99999.999** or **FAUTO, FU****Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min while milling

Input: **0...99999.999** or **FAUTO**

## Example

11 CYCL DEF 265 HEL. THREAD DRLG/MLG ~	
Q335=+5	;NOMINAL DIAMETER ~
Q239=+1	;THREAD PITCH ~
Q201=-18	;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=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q254=+200	;F COUNTERBORING ~
Q207=+500	;FEED RATE MILLING
12 CYCL CALL	



## 5.10 Cycle 267 OUTSIDE THREAD MLLNG (option 19)

ISO programming  
G267

### Application



This function must be enabled and adapted by the machine manufacturer.

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

### Cycle sequence

- 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

## Notes

### 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

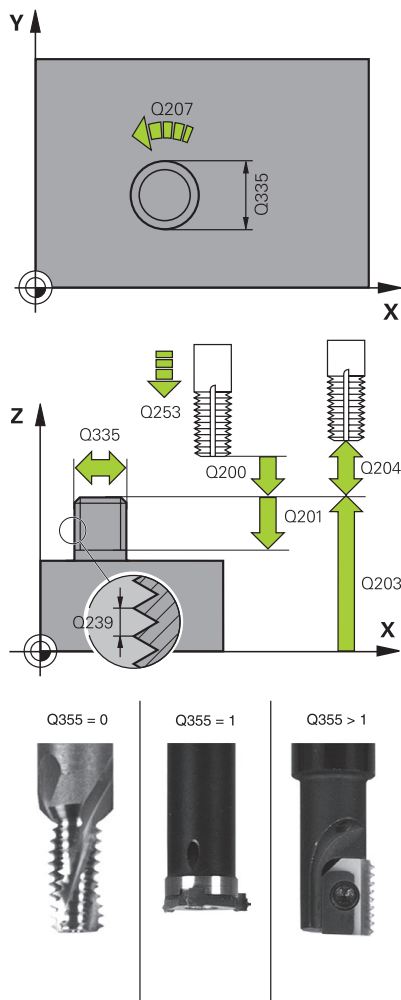
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).
- The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
  - 1 Depth of thread
  - 2 Depth at front

#### Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

## Cycle parameters

### Help graphic



### Parameter

#### Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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: **-99.9999...+99.9999**

#### Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q355 Number of threads per step?

Number of thread revolutions by which the tool is moved:

**0** = one helical line to the thread depth

**1** = continuous helical path over the entire length of the thread

**>1** = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch.

Input: **0...99999**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### 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)

Input: **-1, 0, +1** or **PREDEF**

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

## Help graphic

## Parameter

**Q358 Sinking depth at front?**

Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q359 Countersinking offset at front?**

Distance by which the control moves the tool center away from the center. This value has an incremental effect.

Input: **0...99999.9999**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q254 Feed rate for counterboring?**

Traversing speed of the tool in mm/min during counterboring

Input: **0...99999.999** or **FAUTO, FU**

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min while milling

Input: **0...99999.999** or **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: **0...99999.999** or **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

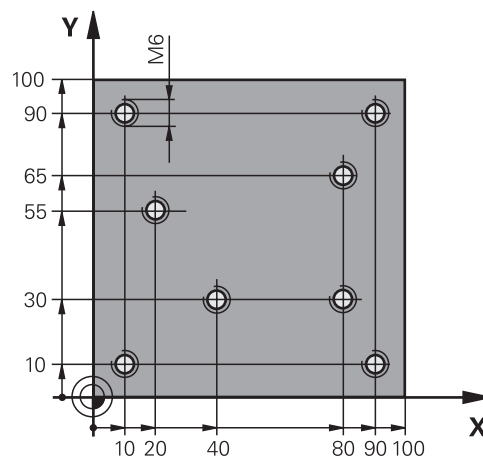
### Example: Thread milling

The drill hole coordinates are stored in LBL 1 and are called by the control with **CALL LBL**.

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 TAP 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 171 Z S5000	; Tool call: centering tool
4 L Z+100 R0 FMAX M3	; Move tool to clearance height (program a value for F): the control positions the tool at the clearance height after every cycle
5 CYCL DEF 240 CENTERING ~	; Cycle definition: Centering
Q200=+2 ;SET-UP CLEARANCE ~	
Q343=+1 ;SELECT DIA./DEPTH ~	
Q201=-1 ;DEPTH ~	
Q344=-7 ;DIAMETER ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE	
6 CALL LBL 1	
7 L Z+100 R0 FMAX	; Retract the tool
8 TOOL CALL 227 Z S5000	; Tool call: drill
9 L Z+100 R0 FMAX M3	; Move tool to clearance height (enter a value for F)
10 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 ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	

Q395=+0	;DEPTH REFERENCE	
11 CALL LBL 1		
12 L Z+100 R0 FMAX		; Retract the tool
13 TOOL CALL 263 Z S200		; Tool call: tap
14 L Z+100 R0 FMAX M3		; Move tool to clearance height
15 CYCL DEF 206 TAPPING ~		; Cycle definition: Tapping
Q200=+2	;SET-UP CLEARANCE ~	
Q201=-22	;DEPTH OF THREAD ~	
Q206=+150	;FEED RATE FOR PLNGNG ~	
Q211=+0	;DWELL TIME AT DEPTH ~	
Q203=+0	;SURFACE COORDINATE ~	
Q204=+50	;2ND SET-UP CLEARANCE	
16 CALL LBL 1		
17 L Z+100 R0 FMAX		; Retract the tool, end program
18 M30		
19 LBL 1		
20 L X+10 Y+10 R0 FMAX M99		
21 L X+40 Y+30 R0 FMAX M99		
22 L X+80 Y+30 R0 FMAX M99		
23 L X+90 Y+10 R0 FMAX M99		
24 L X+80 Y+65 R0 FMAX M99		
25 L X+90 Y+90 R0 FMAX M99		
26 L X+10 Y+90 R0 FMAX M99		
27 L X+20 Y+55 R0 FMAX M99		
28 LBL 0		
29 END PGM TAP MM		













# 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
	Cycle 251 RECTANGULAR POCKET (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Plunging strategy: helical, reciprocating, or vertical</li> </ul>	163
	Cycle 252 CIRCULAR POCKET (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Plunging strategy: helical or vertical</li> </ul>	170
	Cycle 253 SLOT MILLING (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Plunging strategy: reciprocating or vertical</li> </ul>	177
	Cycle 254 CIRCULAR SLOT (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Plunging strategy: reciprocating or vertical</li> </ul>	183
	Cycle 256 RECTANGULAR STUD (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Approach position: selectable</li> </ul>	189
	Cycle 257 CIRCULAR STUD (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Input of the start angle</li> <li>■ Helical infeed starting from the workpiece blank diameter</li> </ul>	195
	Cycle 258 POLYGON STUD (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Helical infeed starting from the workpiece blank diameter</li> </ul>	200
	Cycle 233 FACE MILLING (option 19) <ul style="list-style-type: none"> <li>■ Roughing and finishing cycle</li> <li>■ Roughing strategy and direction: selectable</li> <li>■ Input of side walls</li> </ul>	206

## 6.2 Cycle 251 RECTANGULAR POCKET (option 19)

### ISO programming G251

#### 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 sequence

##### 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

#### Notes

##### 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.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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.
- 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.

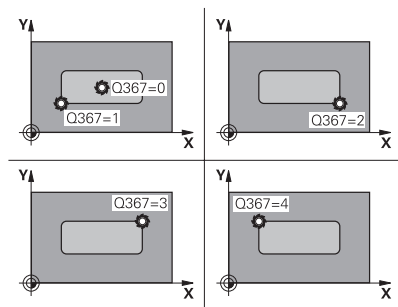
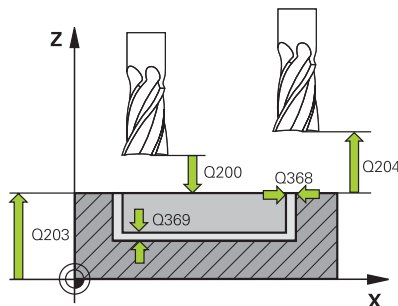
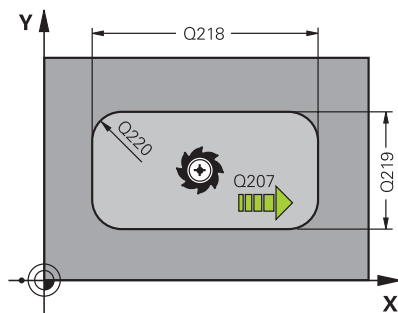
**Further information:** "Plunging strategy Q366 with RCUTS",  
Page 169

**Notes on programming**

- 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 working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- 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.
- Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

## Cycle parameters

### Help graphic



### Parameter

#### Q215 Machining operation (0/1/2)?

Define the machining operation:

**0:** Roughing and finishing

**1:** Only roughing

**2:** Only finishing

Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined

Input: **0, 1, 2**

#### Q218 First side length?

Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q219 Second side length?

Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...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: **0...99999.9999**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q224 Angle of rotation?

Angle by which the entire operation is rotated. The center of rotation is the position at which the tool is located when the cycle is called. The value has an absolute effect.

Input: **-360.000...+360.000**

#### Q367 Position of pocket (0/1/2/3/4)?

Position of the pocket with respect to the tool when the cycle is called:

**0:** Tool position = Center of pocket

**1:** Tool position = Lower left corner

**2:** Tool position = Lower right corner

**3:** Tool position = Upper right corner

**4:** Tool position = Upper left corner

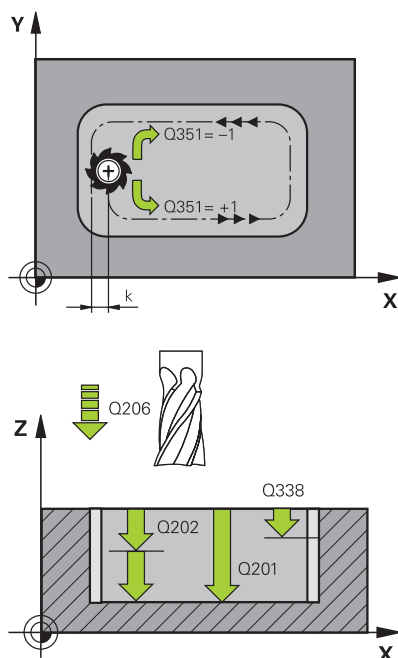
Input: **0, 1, 2, 3, 4**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

## Help graphic



## Parameter

**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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Q201 Depth?**

Distance between workpiece surface and bottom of pocket. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q202 Plunging depth?**

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q206 Feed rate for plunging?**

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q338 Infeed for finishing?**

Tool infeed in the spindle axis per finishing cut.

**Q338 = 0**: Finishing with a single infeed

This value has an incremental effect.

Input: **0...99999.9999**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	<p><b>Q370 Path overlap factor?</b>  <b>Q370</b> x tool radius = stepover factor k.  Input: <b>0.0001...1.41</b> or <b>PREDEF</b></p>
	<p><b>Q366 Plunging strategy (0/1/2)?</b>  Type of plunging strategy:  <b>0:</b> Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle <b>ANGLE</b> defined in the tool table.  <b>1:</b> Helical plunging. In the tool table, the plunging angle <b>ANGLE</b> 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 <b>RCUTS</b> cutting width in the tool table  <b>2:</b> Reciprocating plunge. In the tool table, the plunging angle <b>ANGLE</b> 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 <b>RCUTS</b> cutting width in the tool table  <b>PREDEF:</b> The control uses the value from the GLOBAL DEF block  Input: <b>0, 1, 2</b> or <b>PREDEF</b>  <b>Further information:</b> "Plunging strategy Q366 with RCUTS",  Page 169</p>
	<p><b>Q385 Finishing feed rate?</b>  Traversing speed of the tool in mm/min for side and floor finishing  Input: <b>0...99999.999</b> or <b>FAUTO, FU, FZ</b></p>
	<p><b>Q439 Feed rate reference (0-3)?</b>  Specify the reference for the programmed feed rate:  <b>0:</b> Feed rate is referenced to the path of the tool center  <b>1:</b> Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center  <b>2:</b> Feed rate is referenced to the cutting edge during side finishing <b>and</b> floor finishing; otherwise it is referenced to the path of the tool center  <b>3:</b> Feed rate is always referenced to the cutting edge  Input: <b>0, 1, 2, 3</b></p>

**Example**

11 CYCL DEF 251 RECTANGULAR POCKET ~	
Q215=+0	;MACHINING OPERATION ~
Q218=+60	;FIRST SIDE LENGTH ~
Q219=+20	;2ND SIDE LENGTH ~
Q220=+0	;CORNER RADIUS ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q224=+0	;ANGLE OF ROTATION ~
Q367=+0	;POCKET POSITION ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-20	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q338=+0	;INFED 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
12 L X+50 Y+50 R0 FMAX 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** is, 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

- 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 Cycle 252 CIRCULAR POCKET (option 19)

### ISO programming

G252

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

#### 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.

**Notes****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.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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 176

**Notes on programming**

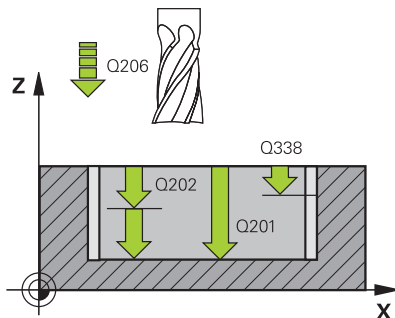
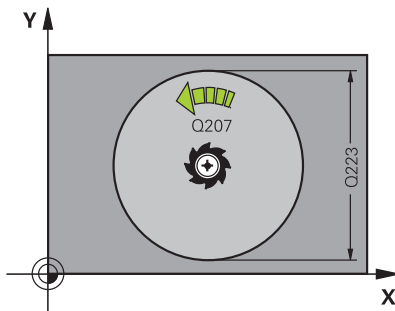
- If the tool table is inactive, you must always plunge vertically (**Q366=0**) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position (circle center) with radius compensation **R0**.
- The algebraic sign for the **DEPTH** cycle parameter determines the working direction. If you program **DEPTH=0**, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

**Note regarding machine parameters**

- For helical plunging, the control will display an error message if the internally calculated helix diameter is less than twice the tool diameter. If you are using a center-cut tool, you can switch this monitoring function off via the **suppressPlungeErr** machine parameter (no. 201006).

## Cycle parameters

### Help graphic



### Parameter

#### Q215 Machining operation (0/1/2)?

Define the machining operation:

**0:** Roughing and finishing

**1:** Only roughing

**2:** Only finishing

Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined

Input: **0, 1, 2**

#### Q223 Circle diameter?

Diameter of the finished pocket

Input: **0...99999.9999**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

#### Q201 Depth?

Distance between workpiece surface and bottom of pocket. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

#### Q369 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

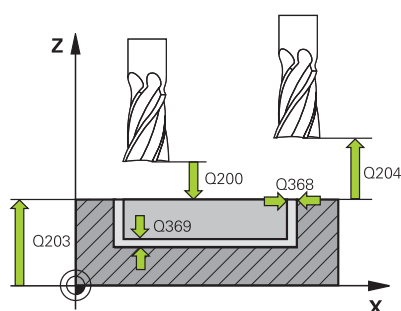
Input: **0...99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO, FU, FZ**

## Help graphic



## Parameter

**Q338 Infeed for finishing?**

Tool infeed in the spindle axis per finishing cut.

**Q338 = 0:** Finishing with a single infeed

This value has an incremental effect.

Input: **0...99999.9999**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **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: **0.1...1999** or **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

Input: **0, 1** or **PREDEF**

**Further information:** "Plunging strategy Q366 with RCUTS",  
Page 176

## Help graphic

## Parameter

**Q385 Finishing feed rate?**

Traversing speed of the tool in mm/min for side and floor finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q439 Feed rate reference (0-3)?**

Specify the reference for the programmed feed rate:

**0:** Feed rate is referenced to the path of the tool center

**1:** Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center

**2:** Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center

**3:** Feed rate is always referenced to the cutting edge

Input: **0, 1, 2, 3**

## Example

11 CYCL DEF 252 CIRCULAR POCKET ~	
Q215=+0	;MACHINING OPERATION ~
Q223=+50	;CIRCLE DIAMETER ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-20	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q338=+0	;INFED 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
12 L X+50 Y+50 R0 FMAX 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** is, 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 Cycle 253 SLOT MILLING (option 19)

### ISO programming

#### G253

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

#### 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.

### Notes

#### 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

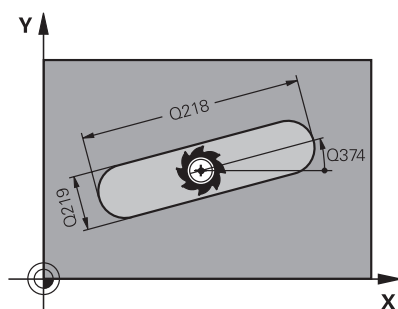
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- If 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.
- 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.

**Notes on programming**

- 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 working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- 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.

## Cycle parameters

### Help graphic



### Parameter

#### Q215 Machining operation (0/1/2)?

Define the machining operation:

**0:** Roughing and finishing

**1:** Only roughing

**2:** Only finishing

Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368, Q369**) has been defined

Input: **0, 1, 2**

#### Q218 Length of slot?

Enter the length of the slot. It is parallel to the main axis of the working plane.

Input: **0...99999.9999**

#### Q219 Width of slot?

Enter the width of the slot. It is in parallel to the secondary axis of the working plane. If you enter a slot width that equals the tool diameter, then the control will carry out the roughing process only (slot milling).

Maximum slot width for roughing: Twice the tool diameter

Input: **0...99999.9999**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q374 Angle of rotation?

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. The value has an absolute effect.

Input: **-360.000...+360.000**

#### Q367 Position of slot (0/1/2/3/4)?

Position of the figure relative to the position of the tool when the cycle is called:

**0:** Tool position = Center of figure

**1:** Tool position = Left end of figure

**2:** Tool position = Center of left figure arc

**3:** Tool position = Center of right figure arc

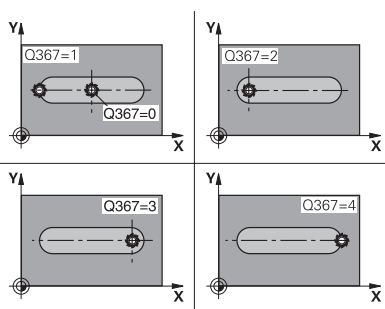
**4:** Tool position = Right end of figure

Input: **0, 1, 2, 3, 4**

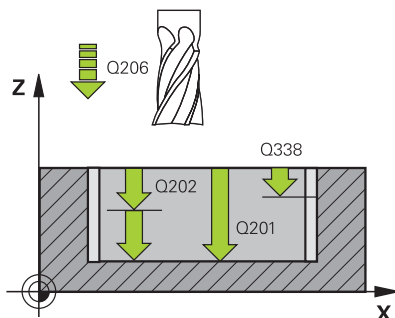
#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**



## Help graphic



## Parameter

**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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Q201 Depth?**

Distance between workpiece surface and slot floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q202 Plunging depth?**

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q206 Feed rate for plunging?**

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q338 Infeed for finishing?**

Tool infeed in the spindle axis per finishing cut.

**Q338 = 0**: Finishing with a single infeed

This value has an incremental effect.

Input: **0...99999.9999**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

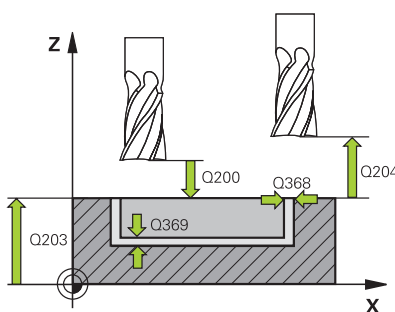
Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**



Help graphic	Parameter
	<p><b>Q366 Plunging strategy (0/1/2)?</b></p> <p>Type of plunging strategy:</p> <p><b>0</b> = Vertical plunging. The plunging angle <b>ANGLE</b> in the tool table is not evaluated.</p> <p><b>1, 2</b> = Reciprocating plunge. In the tool table, the plunging angle <b>ANGLE</b> for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message.</p> <p>Or: <b>PREDEF</b></p> <p>Input: <b>0, 1, 2</b></p>
	<p><b>Q385 Finishing feed rate?</b></p> <p>Traversing speed of the tool in mm/min for side and floor finishing</p> <p>Input: <b>0...99999.999</b> or <b>FAUTO, FU, FZ</b></p>
	<p><b>Q439 Feed rate reference (0-3)?</b></p> <p>Specify the reference for the programmed feed rate:</p> <p><b>0</b>: Feed rate is referenced to the path of the tool center</p> <p><b>1</b>: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center</p> <p><b>2</b>: Feed rate is referenced to the cutting edge during side finishing <b>and</b> floor finishing; otherwise it is referenced to the path of the tool center</p> <p><b>3</b>: Feed rate is always referenced to the cutting edge</p> <p>Input: <b>0, 1, 2, 3</b></p>

**Example**

11 CYCL DEF 253 SLOT MILLING ~	
Q215=+0	;MACHINING OPERATION ~
Q218=+60	;SLOT LENGTH ~
Q219=+10	;SLOT WIDTH ~
Q368=+0	;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	;ALLOWANCE FOR FLOOR ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q338=+0	;INFED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q366=+2	;PLUNGE ~
Q385=+500	;FINISHING FEED RATE ~
Q439=+3	;FEED RATE REFERENCE
12 L X+50 Y+50 R0 FMAX M99	

## 6.5 Cycle 254 CIRCULAR SLOT (option 19)

### ISO programming

#### G254

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

#### 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

### Notes

#### 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.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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 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.
- 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.

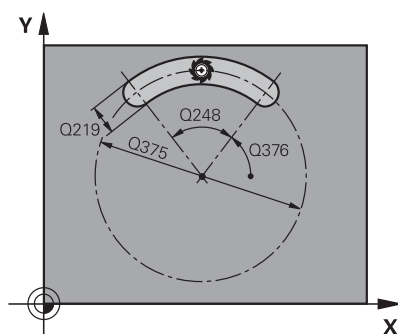
**Notes on programming**

- 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 working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- 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.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.



## Cycle parameters

### Help graphic



### Parameter

#### Q215 Machining operation (0/1/2)?

Define the machining operation:

**0:** Roughing and finishing

**1:** Only roughing

**2:** Only finishing

Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368, Q369**) has been defined

Input: **0, 1, 2**

#### Q219 Width of slot?

Enter the width of the slot. It is in parallel to the secondary axis of the working plane. If you enter a slot width that equals the tool diameter, then the control will carry out the roughing process only (slot milling).

Maximum slot width for roughing: Twice the tool diameter

Input: **0...99999.9999**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q375 Pitch circle diameter?

Enter the diameter of the circle.

Input: **0...99999.9999**

#### Q367 Ref. for slot pos. (0/1/2/3)?

Position of the slot relative to the position of the tool when the cycle is called:

**0:** The tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle.

**1:** Tool position = Center of left slot circle. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account.

**2:** Tool position = Center of center line. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account.

**3:** Tool position = Center of right slot circle. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account.

Input: **0, 1, 2, 3**

#### Q216 Center in 1st axis?

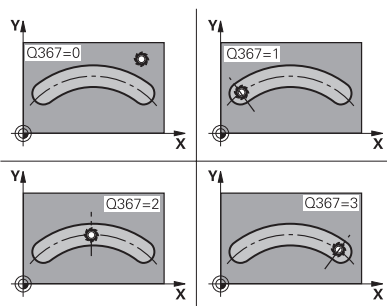
Center of the pitch circle in the main axis of the working plane. **Only effective if Q367 = 0.** The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

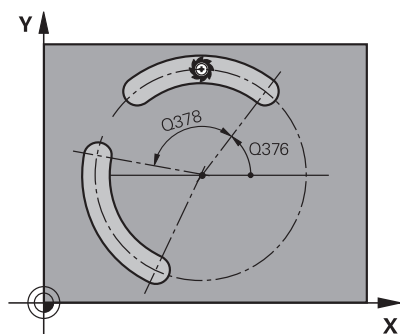
#### Q217 Center in 2nd axis?

Center of the pitch circle in the secondary axis of the working plane. **Only effective if Q367 = 0.** The value has an absolute effect.

Input: **-99999.9999...+99999.9999**



## Help graphic



## Parameter

**Q376 Starting angle?**

Enter the polar angle of the starting point. The value has an absolute effect.

Input: **-360.000...+360.000**

**Q248 Angular length?**

Enter the angular length of the slot. This value has an incremental effect.

Input: **0...360**

**Q378 Intermediate stepping angle?**

Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. This value has an incremental effect.

Input: **-360.000...+360.000**

**Q377 Number of repetitions?**

Number of machining operations on a pitch circle

Input: **1...99999**

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Q201 Depth?**

Distance between workpiece surface and slot floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q202 Plunging depth?**

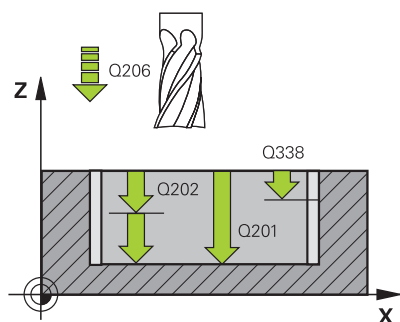
Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

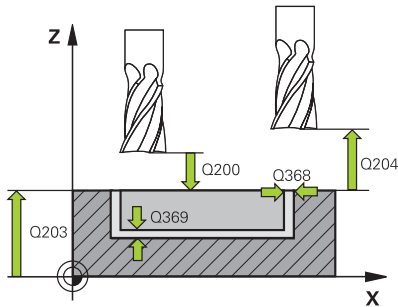
**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**



## Help graphic



## Parameter

**Q206 Feed rate for plunging?**

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q338 Infeed for finishing?**

Tool infeed in the spindle axis per finishing cut.

**Q338 = 0:** Finishing with a single infeed

This value has an incremental effect.

Input: **0...99999.9999**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **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

**PREDEF:** The control uses the value from the GLOBAL DEF block.

Input: **0, 1, 2**

**Q385 Finishing feed rate?**

Traversing speed of the tool in mm/min for side and floor finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

## Help graphic

## Parameter

**Q439 Feed rate reference (0-3)?**

Specify the reference for the programmed feed rate:

**0:** Feed rate is referenced to the path of the tool center

**1:** Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center

**2:** Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center

**3:** Feed rate is always referenced to the cutting edge

Input: **0, 1, 2, 3**

## Example

11 CYCL DEF 254 CIRCULAR SLOT ~	
Q215=+0	;MACHINING OPERATION ~
Q219=+10	;SLOT WIDTH ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q375=+60	;PITCH CIRCLE DIAMETR ~
Q367=+0	;REF. SLOT POSITION ~
Q216=+50	;CENTER IN 1ST AXIS ~
Q217=+50	;CENTER IN 2ND AXIS ~
Q376=+0	;STARTING ANGLE ~
Q248=+0	;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 ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q338=+0	;INFED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q366=+2	;PLUNGE ~
Q385=+500	;FINISHING FEED RATE ~
Q439=+0	;FEED RATE REFERENCE
12 L X+50 Y+50 R0 FMAX M99	

## 6.6 Cycle 256 RECTANGULAR STUD (option 19)

### ISO programming

#### G256

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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

## Notes

### 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.

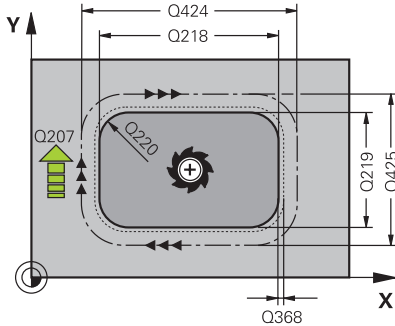
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the 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.

#### Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

#### Q218 First side length?

Length of stud parallel to the main axis of the working plane

Input: 0...99999.9999

#### Q424 Workpiece blank side length 1?

Length of stud blank parallel to the main 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: 0...99999.9999

#### Q219 Second side length?

Length of stud 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: 0...99999.9999

#### Q425 Workpiece blank side length 2?

Length of stud blank parallel to the secondary axis of the working plane.

Input: 0...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: -99999.9999...+99999.9999

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane, is left over after machining. This value has an incremental effect.

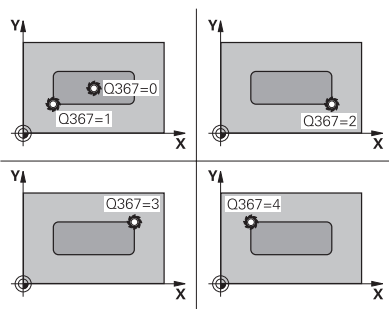
Input: -99999.9999...+99999.9999

#### Q224 Angle of rotation?

Angle by which the entire operation is rotated. The center of rotation is the position at which the tool is located when the cycle is called. The value has an absolute effect.

Input: -360.000...+360.000

## Help graphic



## Parameter

**Q367 Position of stud (0/1/2/3/4)?**

Position of the stud with respect to the tool when the cycle is called.

**0:** Tool position = Center of stud

**1:** Tool position = Lower left corner

**2:** Tool position = Lower right corner

**3:** Tool position = Upper right corner

**4:** Tool position = Upper left corner

Input: **0, 1, 2, 3, 4**

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Q201 Depth?**

Distance between workpiece surface and bottom of stud. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q202 Plunging depth?**

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

**Q206 Feed rate for plunging?**

Traversing speed of the tool in mm/min while moving to depth

Input: **0...99999.999** or **FAUTO, FMAX, FU, FZ**

**Q200 Set-up clearance?**

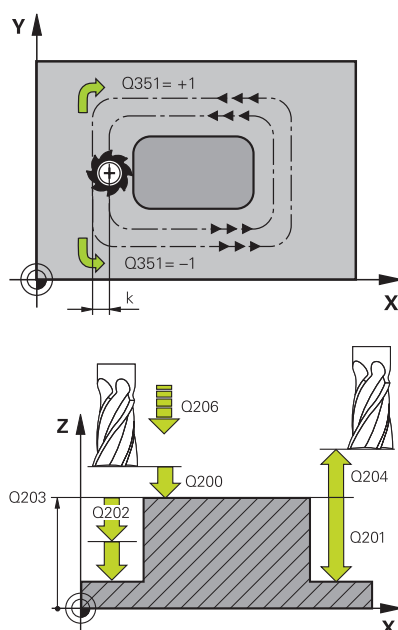
Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**





Help graphic	Parameter
	<b>Q204 2nd set-up clearance?</b> Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: <b>0...99999.9999</b> or <b>PREDEF</b>
	<b>Q370 Path overlap factor?</b> $\text{Q370} \times \text{tool radius} = \text{stepover factor } k$ . Input: <b>0.0001...1.9999</b> or <b>PREDEF</b>
	<b>Q437 Starting position (0...4)?</b> Specify the approach strategy of the tool: <b>0:</b> From the right of the stud (default setting) <b>1:</b> Lower left corner <b>2:</b> Lower right corner <b>3:</b> Upper right corner <b>4:</b> Upper left corner If approach marks appear on the stud surface during approach with the setting <b>Q437=0</b> , then choose another approach position. Input: <b>0, 1, 2, 3, 4</b>
	<b>Q215 Machining operation (0/1/2)?</b> Define the machining operation: <b>0:</b> Roughing and finishing <b>1:</b> Only roughing <b>2:</b> Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance ( <b>Q368, Q369</b> ) has been defined Input: <b>0, 1, 2</b>
	<b>Q369 Finishing allowance for floor?</b> Finishing allowance for the floor. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q338 Infeed for finishing?</b> Tool infeed in the spindle axis per finishing cut. <b>Q338 = 0:</b> Finishing with a single infeed This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q385 Finishing feed rate?</b> Traversing speed of the tool in mm/min for side and floor finishing Input: <b>0...99999.999</b> or <b>FAUTO, FU, FZ</b>

**Example**

11 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+60	;FIRST SIDE LENGTH ~
Q424=+75	;WORKPC. BLANK SIDE 1 ~
Q219=+20	;2ND SIDE LENGTH ~
Q425=+60	;WORKPC. BLANK SIDE 2 ~
Q220=+0	;CORNER RADIUS ~
Q368=+0	;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=+3000	;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	;INFED FOR FINISHING ~
Q385=+500	;FEED RATE FOR FINISHING
12 L X+50 Y+50 R0 FMAX M99	

## 6.7 Cycle 257 CIRCULAR STUD (option 19)

### ISO programming

#### G257

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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

## Notes

### 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.

- ▶ Check the machining sequence using the graphic simulation.

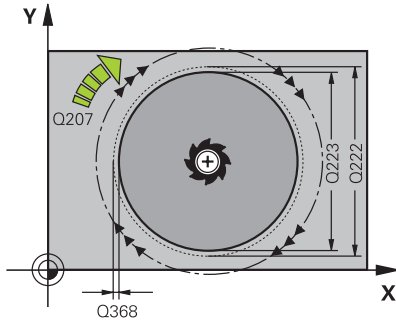
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the 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.

#### Notes on programming

- Pre-position the tool in the working plane to the starting position (stud center) with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

## Cycle parameters

### Help graphic



### Parameter

#### Q223 Finished part diameter?

Diameter of the finished stud

Input: **0...99999.9999**

#### Q222 Workpiece blank diameter?

Diameter of 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: **0...99999.9999**

#### Q368 Finishing allowance for side?

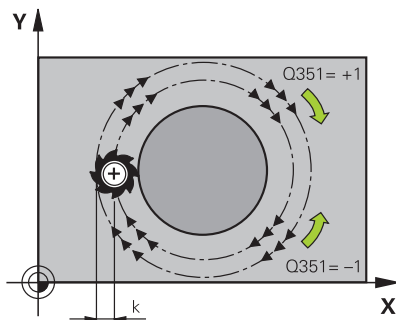
Finishing allowance in the working plane. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**



#### Q351 Direction? Climb=+1, Up-cut=-1

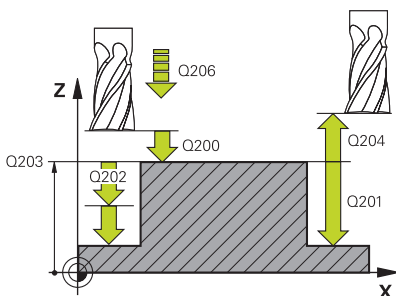
Type of milling operation. The direction of spindle rotation is taken into account.

**+1** = climb milling

**-1** = up-cut milling

**PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**



#### Q201 Depth?

Distance between workpiece surface and bottom of stud. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

#### Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while moving to depth

Input: **0...99999.999** or **FAUTO, FMAX, FU, FZ**

## Help graphic

## Parameter

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q370 Path overlap factor?**

**Q370** x tool radius = stepover factor k.

Input: **0.0001...1.9999** or **PREDEF**

**Q376 Starting angle?**

Polar angle relative to the stud center, from which the tool approaches the stud.

Input: **-1...+359**

**Q215 Machining operation (0/1/2)?**

Specify the machining operation:

**0:** Roughing and finishing

**1:** Only roughing

**2:** Only finishing

Input: **0, 1, 2**

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q338 Infeed for finishing?**

Tool infeed in the spindle axis per finishing cut.

**Q338** = 0: Finishing with a single infeed

This value has an incremental effect.

**Q385 Finishing feed rate?**

Traversing speed of the tool in mm/min for side and floor finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Example**

11 CYCL DEF 257 CIRCULAR STUD ~	
Q223=+50	;FINISHED PART DIA. ~
Q222=+52	;WORKPIECE BLANK DIA. ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-20	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q206=+3000	;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=-1	;STARTING ANGLE ~
Q215=+1	;MACHINING OPERATION ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q338=+0	;INFED FOR FINISHING ~
Q385=+500	;FINISHING FEED RATE
12 L X+50 Y+50 R0 FMAX M99	

## 6.8 Cycle 258 POLYGON STUD (option 19)

### ISO programming

G258

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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

### Notes

#### 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:  $-360^{\circ}$  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, check 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.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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.

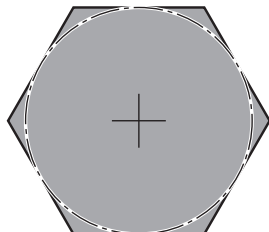
**Notes on programming**

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

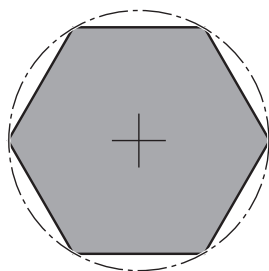
## Cycle parameters

### Help graphic

Q573 = 0



Q573 = 1



### Parameter

#### Q573 Inscr.circle/circumcircle (0/1)?

Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle:

**0:** Dimension is referenced to the inscribed circle

**1:** Dimension is referenced to the circumcircle

Input: **0, 1**

#### Q571 Reference circle diameter?

Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed.

Input: **0...99999.9999**

#### Q222 Workpiece blank diameter?

Enter the diameter of the 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: **0...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: **3...30**

#### Q224 Angle of rotation?

Specify which angle is used to machine the first corner of the polygon stud.

Input: **-360.000...+360.000**

#### 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: **-99999.9999...+99999.9999**

#### Q368 Finishing allowance for side?

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. This value has an incremental effect.

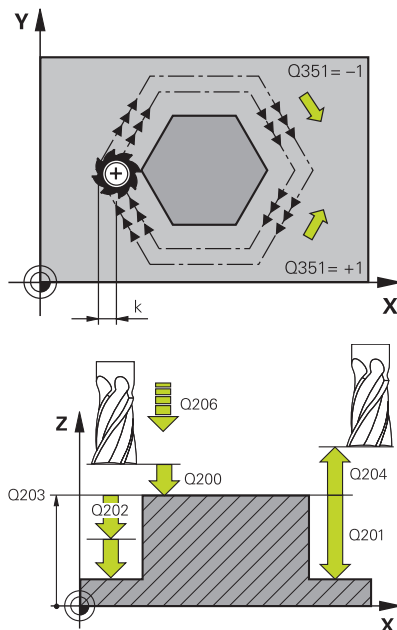
Input: **-99999.9999...+99999.9999**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

## Help graphic



## Parameter

**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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Q201 Depth?**

Distance between workpiece surface and bottom of stud. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q202 Plunging depth?**

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

**Q206 Feed rate for plunging?**

Traversing speed of the tool in mm/min while moving to depth

Input: **0...99999.999** or **FAUTO, FMAX, FU, FZ**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q370 Path overlap factor?**

**Q370** x tool radius = stepover factor k.

Input: **0.0001...1.9999** or **PREDEF**

Help graphic	Parameter
	<b>Q215 Machining operation (0/1/2)?</b> Define the machining operation: <b>0:</b> Roughing and finishing <b>1:</b> Only roughing <b>2:</b> Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance ( <b>Q368, Q369</b> ) has been defined Input: <b>0, 1, 2</b>
	<b>Q369 Finishing allowance for floor?</b> Finishing allowance for the floor. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q338 Infeed for finishing?</b> Tool infeed in the spindle axis per finishing cut. <b>Q338 = 0:</b> Finishing with a single infeed This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q385 Finishing feed rate?</b> Traversing speed of the tool in mm/min for side and floor finishing Input: <b>0...99999.999</b> or <b>FAUTO, FU, FZ</b>

**Example**

11 CYCL DEF 258 POLYGON STUD ~	
Q573=+0	;REFERENCE CIRCLE ~
Q571=+50	;REF-CIRCLE DIAMETER ~
Q222=+52	;WORKPIECE BLANK DIA. ~
Q572=+6	;NUMBER OF CORNERS ~
Q224=+0	;ANGLE OF ROTATION ~
Q220=+0	;RADIUS / CHAMFER ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-20	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q206=+3000	;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
12 L X+50 Y+50 R0 FMAX M99	

## 6.9 Cycle 233 FACE MILLING (option 19)

### ISO programming

G233

### Application



This function must be enabled and adapted by the machine manufacturer.

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:

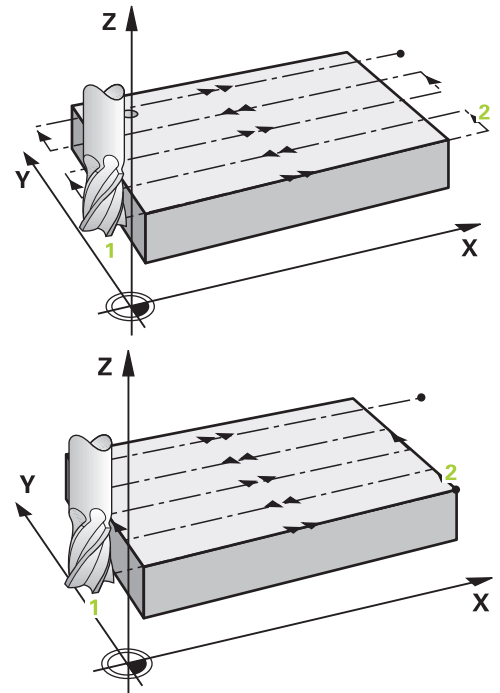
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1:** Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** The surface is machined line by line with overtravel; stepover when retracting at rapid traverse
- **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

**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.

**Cycle sequence**

- 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.
- 4 The control moves the tool to end point **2** at the programmed feed rate for milling.
- 5 The control then shifts the tool laterally to the starting point of the next line at the pre-positioning feed rate. 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 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**.

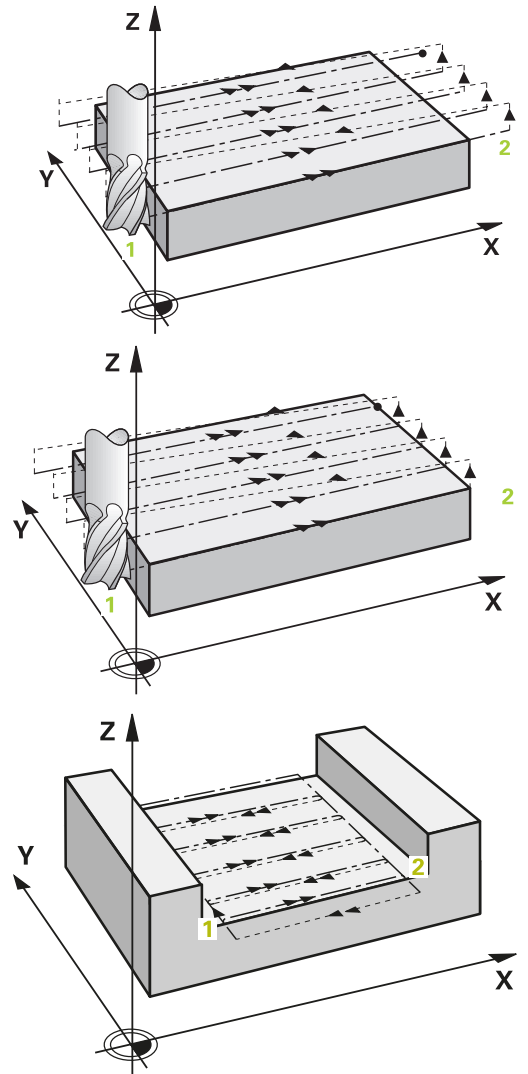


### Strategies Q389=2 and Q389 =3

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.

#### Cycle sequence

- 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.
- 4 The tool subsequently advances at the programmed feed rate for milling to the end point **2**.
- 5 The control positions the tool in the tool 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 the 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 spindle 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**.





**Strategies Q389=2 and Q389=3**

If you program a lateral limitation, the control might not be able to perform movements outside of the contour. In this case the cycle sequences as follows:

- 1 The control positions the tool at **FMAX** to the starting point in the working plane. This position is offset from the edge of the workpiece by the tool radius and the set-up clearance **Q357** to the side.
- 2 The tool moves at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** and from there at **Q207 FEED RATE MILLING** to the first plunging depth **Q202**.
- 3 The control moves the tool on a circular path to the starting point point **1**.
- 4 The tool moves at the programmed feed rate **Q207** to the end point **2** and departs from the contour on a circular path.
- 5 Then the control positions the tool at **Q253 F PRE-POSITIONING** to the approach position of the next path.
- 6 Steps 3 to 5 are repeated until the entire surface is milled.
- 7 If more than one infeed depth is programmed, the control moves the tool at the end of the last path to the set-up clearance **Q200** and positions in the working plane to the next approach position.
- 8 In the last infeed the control mills the **Q369 ALLOWANCE FOR FLOOR** at **Q385 FINISHING FEED RATE**.
- 9 At the end of the last path, the control retracts the tool to the 2nd set-up clearance **Q204** and then to the position last programmed before the cycle.



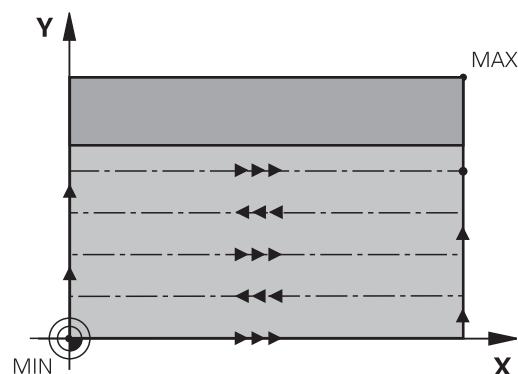
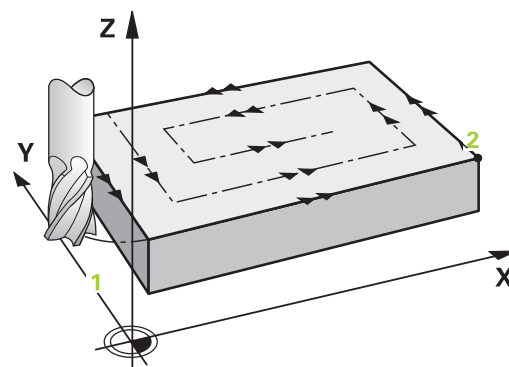
- The circular paths for approaching and departing the paths depend on **Q220 CORNER RADIUS**.
- The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor **Q370** and the set-up clearance to the side **Q357**.

**Strategy Q389=4****Cycle sequence**

- 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.
- 4 The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a tangential approach path.
- 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 completed. 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 spindle 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.



## Notes

### 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.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- Cycle **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.
- 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.

#### Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- 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).
- If you define **Q370 TOOL PATH OVERLAP** >1, the programmed overlap factor will be taken into account right from the first machining path.
- 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.

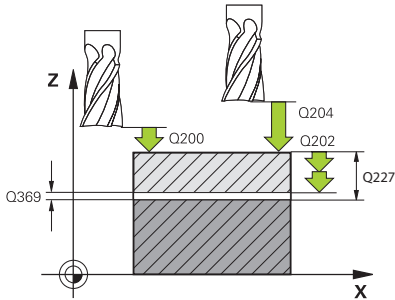


Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

## Cycle parameters

Help graphic	Parameter
	<b>Q215 Machining operation (0/1/2)?</b> Define the machining operation: <b>0:</b> Roughing and finishing <b>1:</b> Only roughing <b>2:</b> Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance ( <b>Q368, Q369</b> ) has been defined Input: <b>0, 1, 2</b>
	<b>Q389 Machining strategy (0-4)?</b> Specify how the control machines the surface: <b>0:</b> Meander machining, stepover at positioning feed rate outside the surface to be machined <b>1:</b> Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined <b>2:</b> Machining line by line, retraction and stepover at positioning feed rate outside the surface to be machined <b>3:</b> Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined <b>4:</b> Helical machining, uniform infeed from the outside toward the inside Input: <b>0, 1, 2, 3, 4</b>
	<b>Q350 Milling direction?</b> Axis in the working plane that defines the machining direction: <b>1:</b> Main axis = Machining direction <b>2:</b> Secondary axis = Machining direction Input: <b>1, 2</b>
	<b>Q218 First side length?</b> Length of the surface to be machined in the main axis of the working plane, referencing the starting point in the 1st axis. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q219 Second side length?</b> 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 referenced to the <b>STARTNG PNT 2ND AXIS</b> . This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>

## Help graphic



## Parameter

**Q227 Starting point in 3rd axis?**

Coordinate of the workpiece surface used to calculate the infeeds. The value has an absolute effect.

Input: -99999.9999...+99999.9999

**Q386 End point in 3rd axis?**

Coordinate in the spindle axis on which the surface will be face-milled. The value has an absolute effect.

Input: -99999.9999...+99999.9999

**Q369 Finishing allowance for floor?**

Value used for the last infeed. This value has an incremental effect.

Input: 0...99999.9999

**Q202 Maximum plunging depth?**

Infeed per cut. Enter an incremental value greater than 0.

Input: 0...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: 0.0001...1.9999

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min for milling

Input: 0...99999.999 or FAUTO, FU, FZ

**Q385 Finishing feed rate?**

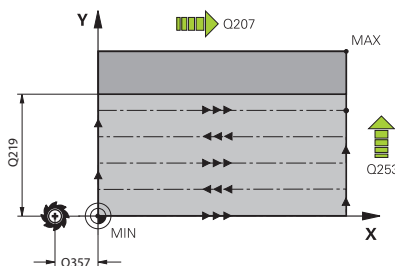
Traversing speed of the tool in mm/min while milling the last infeed

Input: 0...99999.999 or 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: 0...99999.9999 or FMAX, FAUTO, PREDEF



## Help graphic

## Parameter

**Q357 Safety clearance to the side?**

Parameter **Q357** influences the following situations:

**Approaching the first infeed depth:** **Q357** is the lateral distance from the tool to the workpiece.

**Roughing with the Q389 = 0 to 3 roughing strategies:** The surface to be machined is extended in **Q350 MILLING DIRECTION** by the value from **Q357** if no limit has been set in that direction.

**Side finishing:** The paths are extended by **Q357** in the **Q350 MILLING DIRECTION**.

This value has an incremental effect.

Input: **0...99999.9999**

**Q200 Set-up clearance?**

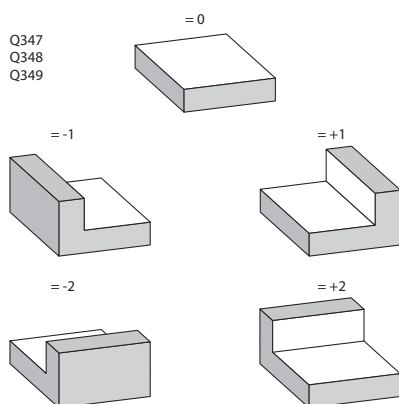
Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**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:

**0:** No limitation

**-1:** Limit in negative main axis

**+1:** Limit in positive main axis

**-2:** Limit in negative secondary axis

**+2:** Limit in positive secondary axis

Input: **-2, -1, 0, +1, +2**

**Q348 2nd limit?**

See parameter **Q347** 1st limit

Input: **-2, -1, 0, +1, +2**

**Q349 3rd limit?**

See parameter **Q347** 1st limit

Input: **-2, -1, 0, +1, +2**

**Q220 Corner radius?**

Radius of a corner at limits (**Q347** to **Q349**)

Input: **0...99999.9999**

Help graphic	Parameter
	<b>Q368 Finishing allowance for side?</b> Finishing allowance in the working plane. This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q338 Infeed for finishing?</b> Tool infeed in the spindle axis per finishing cut. <b>Q338 = 0:</b> Finishing with a single infeed This value has an incremental effect. Input: <b>0...99999.9999</b>
	<b>Q367 Surface position (-1/0/1/2/3/4)?</b> Position of the surface relative to the position of the tool when the cycle is called: <b>-1:</b> Tool position = Current position <b>0:</b> Tool position = Center of stud <b>1:</b> Tool position = Lower left corner <b>2:</b> Tool position = Lower right corner <b>3:</b> Tool position = Upper right corner <b>4:</b> Tool position = Upper left corner Input: <b>-1, 0, +1, +2, +3, +4</b>

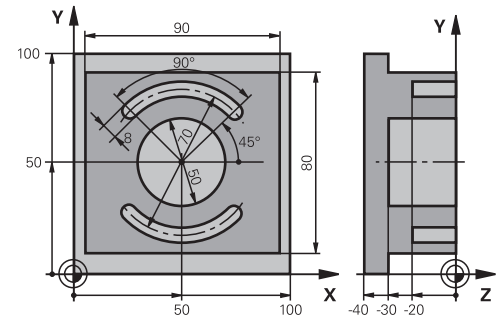
**Example**

11 CYCL DEF 233 FACE MILLING ~	
Q215=+0	;MACHINING OPERATION ~
Q389=+2	;MILLING STRATEGY ~
Q350=+1	;MILLING DIRECTION ~
Q218=+60	;FIRST SIDE LENGTH ~
Q219=+20	;2ND SIDE LENGTH ~
Q227=+0	;STARTNG PNT 3RD AXIS ~
Q386=+0	;END POINT 3RD AXIS ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q202=+5	;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=+0	;CORNER RADIUS ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q338=+0	;INFEEED FOR FINISHING ~
Q367=-1	;SURFACE POSITION
12 L X+50 Y+50 R0 FMAX M99	



## 6.10 Programming Examples

### Example: Milling pockets, studs and slots



0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 6 Z S3500	; Tool call: roughing/finishing
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD ~	
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=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+150 ;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 ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q338=+10 ;INFEEED FOR FINISHING ~	
Q385=+500 ;FINISHING FEED RATE	
6 L X+50 Y+50 R0 FMAX M99	; Cycle call for outside machining
7 CYCL DEF 252 CIRCULAR POCKET ~	
Q215=+0 ;MACHINING OPERATION ~	
Q223=+50 ;CIRCLE DIAMETER ~	

Q368=+0.2	;ALLOWANCE FOR SIDE ~	
Q207=+500	;FEED RATE MILLING ~	
Q351=+1	;CLIMB OR UP-CUT ~	
Q201=-30	;DEPTH ~	
Q202=+5	;PLUNGING DEPTH ~	
Q369=+0.1	;ALLOWANCE FOR FLOOR ~	
Q206=+150	;FEED RATE FOR PLNGNG ~	
Q338=+5	;INFEED FOR FINISHING ~	
Q200=+2	;SET-UP CLEARANCE ~	
Q203=+0	;SURFACE COORDINATE ~	
Q204=+50	;2ND SET-UP CLEARANCE ~	
Q370=+1	;TOOL PATH OVERLAP ~	
Q366=+1	;PLUNGE ~	
Q385=+750	;FINISHING FEED RATE ~	
Q439=+0	;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99		; Cycle call for circular pocket
9 TOOL CALL 3 Z S5000		; Tool call: slot milling cutter
10 L Z+100 R0 FMAX M3		
11 CYCL DEF 254 CIRCULAR SLOT ~		
Q215=+0	;MACHINING OPERATION ~	
Q219=+8	;SLOT WIDTH ~	
Q368=+0.2	;ALLOWANCE FOR SIDE ~	
Q375=+70	;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=+180	;STEPPING ANGLE ~	
Q377=+2	;NR OF REPETITIONS ~	
Q207=+500	;FEED RATE MILLING ~	
Q351=+1	;CLIMB OR UP-CUT ~	
Q201=-20	;DEPTH ~	
Q202=+5	;PLUNGING DEPTH ~	
Q369=+0.1	;ALLOWANCE FOR FLOOR ~	
Q206=+150	;FEED RATE FOR PLNGNG ~	
Q338=+5	;INFEED FOR FINISHING ~	
Q200=+2	;SET-UP CLEARANCE ~	
Q203=+0	;SURFACE COORDINATE ~	
Q204=+50	;2ND SET-UP CLEARANCE ~	
Q366=+2	;PLUNGE ~	
Q385=+500	;FINISHING FEED RATE ~	
Q439=+0	;FEED RATE REFERENCE	

<b>12 CYCL CALL</b>	; Cycle call for slots
<b>13 L Z+100 R0 FMAX</b>	; Retract the tool, end program
<b>14 M30</b>	
<b>15 END PGM C210 MM</b>	










# 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
	Cycle 7 DATUM SHIFT <ul style="list-style-type: none"> <li>■ Shifting contours directly in the NC program</li> <li>■ Or shifting contours using datum tables</li> </ul>	223
	Cycle 8 MIRRORING <ul style="list-style-type: none"> <li>■ Mirroring contours</li> </ul>	226
	Cycle 10 ROTATION <ul style="list-style-type: none"> <li>■ Rotating contours in the working plane</li> </ul>	227
	Cycle 11 SCALING FACTOR <ul style="list-style-type: none"> <li>■ Resizing contours</li> </ul>	229
	Cycle 26 AXIS-SPECIFIC SCALING <ul style="list-style-type: none"> <li>■ Axis-specific resizing of contours</li> </ul>	230
	Cycle 19 WORKING PLANE (option 8) <ul style="list-style-type: none"> <li>■ Executing machining operations in a tilted coordinate system</li> <li>■ On machines with swivel heads and/or rotary tables</li> </ul>	231
	Cycle 247 PRESETTING <ul style="list-style-type: none"> <li>■ Datum setting during program run</li> </ul>	237

### 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 Cycle 7 DATUM SHIFT

### ISO programming

#### G54

### Application



Refer to your machine manual.

A datum shift allows machining operations to be repeated at various locations on the workpiece. Within an NC program, you can either program datum points directly in the cycle definition or call them from a datum table.

You use datum tables for the following purposes:

- Frequent use of the same datum shift
- Frequently recurring machining sequences on different workpieces
- Frequently recurring machining sequences at various locations on one 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.

### Status display

The additional status display **TRANS** contains the following information:

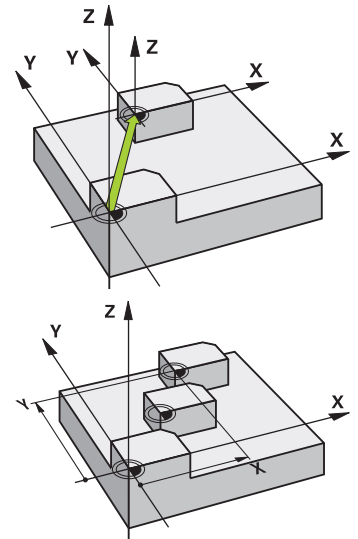
- Coordinates from the datum shift
- Name and path of the active datum table
- Active datum number for datum tables
- Comment from the **DOC** column of the active datum number from the datum table

### Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining modes.
- The main axis, secondary axis and tool axis are in effect in the W-CS or WPL-CS coordinate system. Rotary axes and parallel axes are in effect in the M-CS system.

### Notes about machine parameters

- In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.



**Additional information regarding datum shifts with datum tables:**

- 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.



## Cycle parameters

### Datum shift without a datum table

Help graphic	Parameter
	<b>Displacement?</b> Enter the coordinates of the new datum. Absolute values are referenced to the workpiece datum, which is determined by the presetting. Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted). Up to six NC axes are possible. Input: <b>-999999999...+999999999</b>

#### Example

```

11 CYCL DEF 7.0 DATUM SHIFT
12 CYCL DEF 7.1 X+60
13 CYCL DEF 7.2 Y+40
14 CYCL DEF 7.3 Z+5

```

### Datum shift with a datum table

Help graphic	Parameter
	<b>Displacement?</b> Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input: <b>0...9999</b>

#### Example

```

11 CYCL DEF 7.0 DATUM SHIFT
12 CYCL DEF 7.1 #5

```

## 7.3 Cycle 8 MIRRORING

### ISO programming

#### G28

### Application

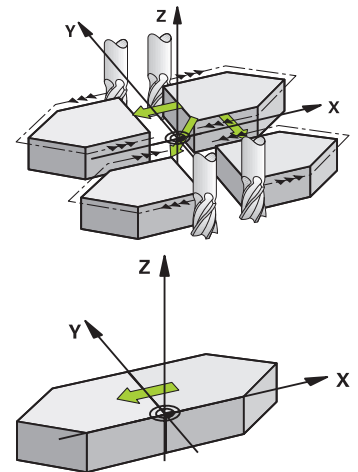
The control can machine the mirror image of a contour in the working plane.

Mirroring becomes effective as soon as it has been defined in the NC program. It is also in effect 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**.

### Notes

- 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

Help graphic	Parameter
	<b>Mirror image axis?</b>
	Enter the axis to be mirrored. You can mirror all axes—including rotary axes—with the exception of the spindle axis and its associated secondary axis. You can enter up to three NC axes.
	Input: <b>X, Y, Z, U, V, W, A, B, C</b>

### Example

```
11 CYCL DEF 8.0 MIRRORING
```

```
12 CYCL DEF 8.1 X Y Z
```

## 7.4 Cycle 10 ROTATION

### ISO programming

#### G73

### Application

Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

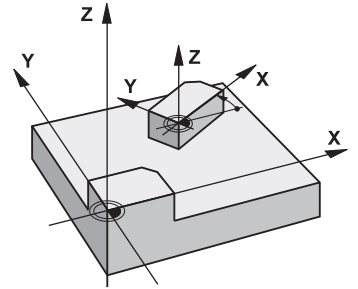
The ROTATION cycle becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Positioning w/ Manual Data Input** operating mode. The active angle of rotation is shown in the additional status display.

#### 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°.

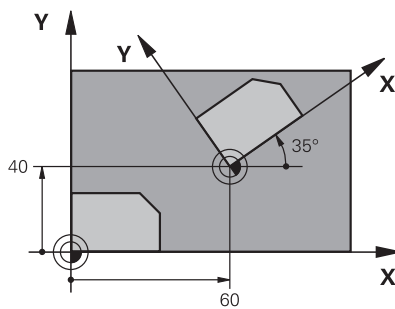


## Notes

- 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

### Help graphic



### Parameter

#### Rotation angle?

Enter the angle of rotation in degrees (°). Enter the value as an incremental or absolute value.

Input: **-360.000...+360.000**

### Example

```
11 CYCL DEF 10.0 ROTATION
```

```
12 CYCL DEF 10.1 ROT+35
```

## 7.5 Cycle 11 SCALING FACTOR

### ISO programming

#### 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 in effect 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

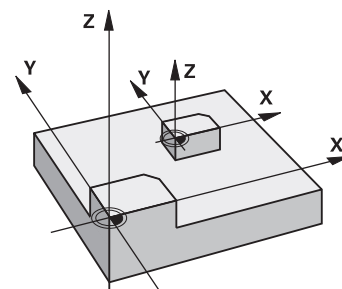
- all three coordinate axes at the same time
- dimensions in cycles

### Requirement

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)



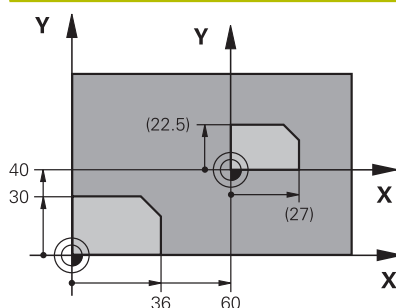
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

### Reset

Program Cycle **11 SCALING FACTOR** again and specify a scaling factor of 1.

### Cycle parameters

#### Help graphic



#### Parameter

##### Factor?

Enter the scaling factor SCL. The control multiplies the coordinates and radii by the SCL factor (as described under "Effect" above).

Input: **0.000001...99.999999**

### Example

```
11 CYCL DEF 11.0 SCALING FACTOR
```

```
12 CYCL DEF 11.1 SCL 0.75
```

## 7.6 Cycle 26 AXIS-SPECIFIC SCALING

### ISO programming

NC syntax is available only in Klartext programming.

### 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 in effect 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.

### Notes

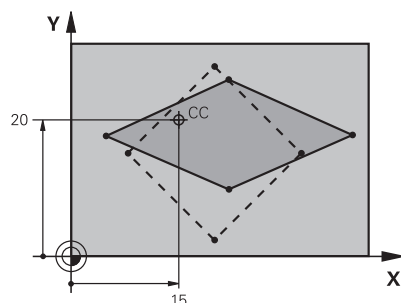
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The contour is enlarged or reduced relative to the center, and not necessarily (as in Cycle **11 SCALING FACTOR**) relative to the active datum.

### Notes on programming

- Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.
- You can program each coordinate axis with its own axis-specific scaling factor.
- In addition, you can enter the coordinates of a center for all scaling factors.

### Cycle parameters

#### Help graphic



#### Parameter

##### Axis and factor?

Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction.

Input: **0.000001...99.999999**

##### Centerpoint coord. of extension?

Center of the axis-specific enlargement or reduction.

Input: **-999999999...+999999999**

### Example

```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
```

```
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

## 7.7 Cycle 19 WORKING PLANE (option 8)

### ISO programming

G80

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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.



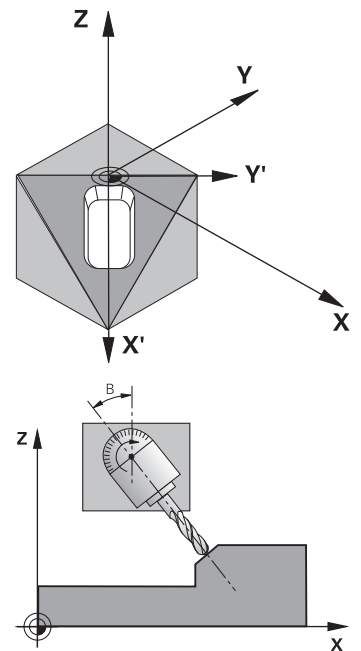
Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

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 have set the **Tilting program run** function to **Active** in the Manual Operation operating mode, the angular value entered in this menu is overwritten by Cycle **19 WORKING PLANE**.



## Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- In combination with a radial facing slide kinematics model, 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.

## Notes on programming

- 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.



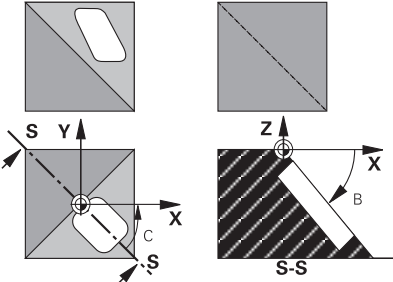
Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

## Notes about machine parameters

- The machine manufacturer 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 manufacturer defines the coordinate system in which the status display shows an active datum shift.



## Cycle parameters

Help graphic	Parameter
	<b>Rotary axis and angle?</b> Enter the axis of rotation together with the associated tilt angles. Program the rotary axes A, B and C using soft keys. Input: <b>-360.000...+360.000</b>

If the control automatically positions the rotary axes, you can enter the following parameters:

Help graphic	Parameter
	<b>Feed rate? F=</b> Traverse speed of the rotary axis during automatic positioning Input: 0...300000
	<b>Set-up clearance?</b> 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. This value has an incremental effect. Input: <b>0...999999999</b>

## 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 manufacturer 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

11 L Z+100 R0 FMAX	
12 L X+25 Y+10 R0 FMAX	
* - ...	; Define the spatial angles for calculating the compensation
13 CYCL DEF 19.0 WORKING PLANE	
14 CYCL DEF 19.1 A+0 B+45 C+0	
15 L A+Q120 C+Q122 R0 F1000	; Position the rotary axes by using values calculated by Cycle 19
16 L Z+80 R0 FMAX	; Activate compensation for the spindle axis
17 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

11 L Z+100 R0 FMAX	
12 L X+25 Y+10 R0 FMAX	
* - ...	; Angle for calculating the compensation; define the feed rate and clearance
13 CYCL DEF 19.0 WORKING PLANE	
14 CYCL DEF 19.1 A+0 B+45 C+0 F5000 ABST50	
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

## 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 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, reverse the 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

## 7.8 Cycle 247 PRESETTING

### ISO programming

#### G247

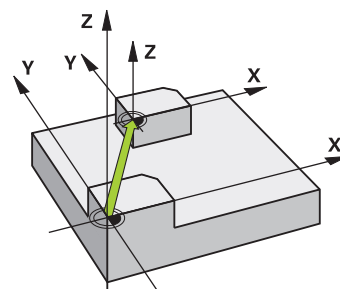
### Application

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.



### Notes

- 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 in effect in the Test Run operating mode.

### Cycle parameters

Help graphic	Parameter
	<b>Number for preset?</b>
	Enter the number of the desired preset from the preset table. Alternatively, you can use the <b>SELECT</b> soft key to directly select the desired preset from the preset table.
	Input: <b>0...65535</b>

### Example

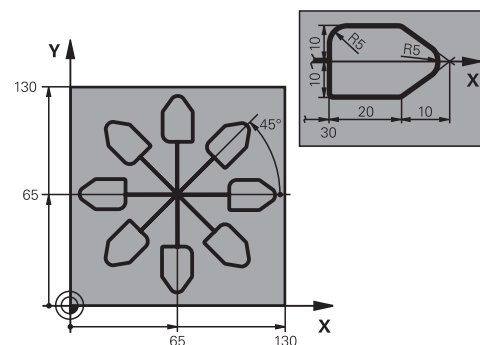
11 CYCL DEF 247 PRESETTING ~
Q339=+4 ;PRESET NUMBER

## 7.9 Programming Examples

### Example: coordinate transformation cycles

#### Program sequence

- Program the coordinate transformations in the main program
- Machining within a subprogram



0 BEGIN PGM C220 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+130 Y+130 Z+0	
3 TOOL CALL 1 Z S4500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 TRANS DATUM AXIS X+65 Y+65	; Shift datum to center
6 CALL LBL 1	; Call milling operation
7 LBL 10	; Set label for program-section repeat
8 CYCL DEF 10.0 ROTATION	
9 CYCL DEF 10.1 IROT+45	
10 CALL LBL 1	; Call milling operation
11 CALL LBL 10 REP6	; Jump back to LBL 10; repeat six times
12 CYCL DEF 10.0 ROTATION	
13 CYCL DEF 10.1 ROT+0	
14 TRANS DATUM RESET	; Reset datum shift
15 L Z+250 R0 FMAX	; Retract the tool
16 M30	; End program
17 LBL 1	; Subprogram 1
18 L X+0 Y+0 R0 FMAX	; Define milling operation
19 L Z+2 R0 FMAX	
20 L Z-5 R0 F200	
21 L X+30 RL	
22 L IY+10	
23 RND R5	
24 L IX+20	
25 L IX+10 IY-10	
26 RND R5	
27 L IX-10 IY-10	
28 L IX-10 IY-10	
29 L IX-20	
30 L IY+10	

31 L X+0 Y+0 R0 F5000	
32 L Z+20 R0 FMAX	
33 LBL 0	
34 END PGM C220 MM	








# 8

**Cycles:  
Pattern Definitions**

## 8.1 Fundamentals

### Overview

The control provides three cycles for machining point patterns:

Soft key	Cycle	Page
	Cycle 220 POLAR PATTERN (option 19) <ul style="list-style-type: none"> <li>■ Defining a circular pattern</li> <li>■ Full circle or pitch circle</li> <li>■ Input of start and end angles</li> </ul>	244
	Cycle 221 CARTESIAN PATTERN (option 19) <ul style="list-style-type: none"> <li>■ Defining a linear pattern</li> <li>■ Input of an angle of rotation</li> </ul>	247
	Cycle 224 DATAMATRIX CODE PATTERN (option 19) <ul style="list-style-type: none"> <li>■ Converting text to a DataMatrix code to be used as a point pattern</li> <li>■ Input of position and size</li> </ul>	251

You can combine the following cycles with point pattern cycles:

	Cycle 220	Cycle 221	Cycle 224
200 DRILLING	✓	✓	✓
201 REAMING	✓	✓	✓
202 BORING	✓	✓	–
203 UNIVERSAL DRILLING	✓	✓	✓
204 BACK BORING	✓	✓	–
205 UNIVERSAL PECKING	✓	✓	✓
206 TAPPING	✓	✓	–
207 RIGID TAPPING	✓	✓	–
208 BORE MILLING	✓	✓	✓
209 TAPPING W/ CHIP BRKG	✓	✓	–
240 CENTERING	✓	✓	✓
251 RECTANGULAR POCKET	✓	✓	✓
252 CIRCULAR POCKET	✓	✓	✓
253 SLOT MILLING	✓	✓	–
254 CIRCULAR SLOT	–	✓	–
256 RECTANGULAR STUD	✓	✓	–
257 CIRCULAR STUD	✓	✓	–
262 THREAD MILLING	✓	✓	–
263 THREAD MLLNG/CNTSNKG	✓	✓	–
264 THREAD DRILLNG/MLLNG	✓	✓	–
265 HEL. THREAD DRLG/MLG	✓	✓	–
267 OUTSIDE THREAD MLLNG	✓	✓	–



If you have to machine irregular point patterns, use **CYCL CALL PAT** to develop point tables.

More regular point patterns are available with the **PATTERN DEF** function.

**Further information:** User's Manual for **Klartext Programming** or **ISO Programming**

**Further information:** "Pattern definition with PATTERN DEF",  
Page 54

## 8.2 Cycle 220 POLAR PATTERN (option 19)

### ISO programming

G220

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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 Single Block mode, the control stops between the individual points of a point pattern.

### Notes

- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

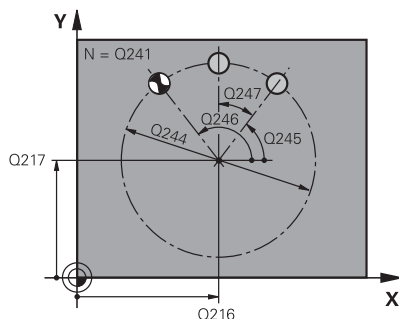
### Note on programming

- If you combine one of the machining cycles **200** to **209** or **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 Cycle **200** is defined in an NC program with **Q203=0** and you then program 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

### Help graphic



### Parameter

#### Q216 Center in 1st axis?

Pitch circle center in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q217 Center in 2nd axis?

Pitch circle center in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q244 Pitch circle diameter?

Diameter of circle

Input: **0...99999.9999**

#### Q245 Starting angle?

Angle between the main axis of the working plane and the starting point for the first machining operation on the pitch circle. The value has an absolute effect.

Input: **-360.000...+360.000**

#### Q246 Stopping angle?

Angle between the main 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. The value has an absolute effect.

Input: **-360.000...+360.000**

#### Q247 Intermediate stepping angle?

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). This value has an incremental effect.

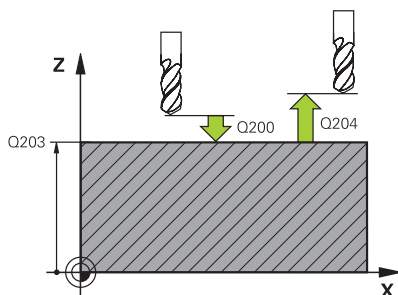
Input: **-360.000...+360.000**

#### Q241 Number of repetitions?

Number of machining operations on a pitch circle

Input: **1...99999**

## Help graphic



## Parameter

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q301 Move to clearance height (0/1)?**

Specify how the tool moves between machining processes:

**0:** Move to the set-up clearance between operations

**1:** Move to the 2nd set-up clearance between operations

Input: **0, 1**

**Q365 Type of traverse? Line=0/arc=1**

Specify how the tool moves between machining processes:

**0:** Move between operations on a straight line

**1:** Move between operations on the pitch circle

Input: **0, 1**

## Example

11 CYCL DEF 220 POLAR PATTERN ~	
Q216=+50	;CENTER IN 1ST AXIS ~
Q217=+50	;CENTER IN 2ND AXIS ~
Q244=+60	;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=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q301=+1	;MOVE TO CLEARANCE ~
Q365=+0	;TYPE OF TRAVERSE
12 CYCL CALL	

## 8.3 Cycle 221 CARTESIAN PATTERN (option 19)

### ISO programming

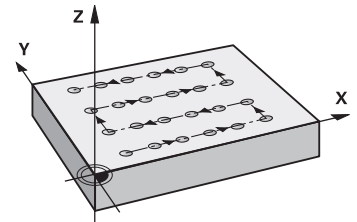
#### G221

### Application



This function must be enabled and adapted by the machine manufacturer.

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



### Cycle sequence

- 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 Single Block mode, the control stops between the individual points of a point pattern.

## Notes

- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

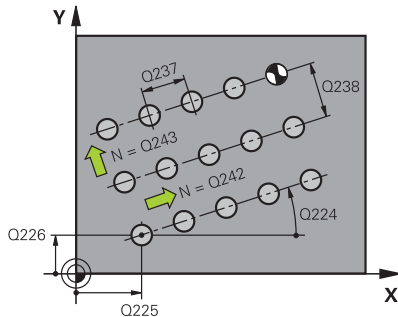
## Notes on programming

- If you combine one of the machining cycles **200** to **209** or **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

### Help graphic



### Parameter

#### Q225 Starting point in 1st axis?

Coordinate of starting point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q226 Starting point in 2nd axis?

Coordinate of starting point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q237 Spacing in 1st axis?

Spacing between the individual points on a line. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q238 Spacing in 2nd axis?

Spacing between the individual lines. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q242 Number of columns?

Number of machining operations on a line

Input: **0...99999**

#### Q243 Number of lines?

Number of lines

Input: **0...99999**

#### Q224 Angle of rotation?

Angle by which the entire pattern is rotated. The center of rotation lies in the starting point. The value has an absolute effect.

Input: **-360.000...+360.000**

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q203 Workpiece surface coordinate?

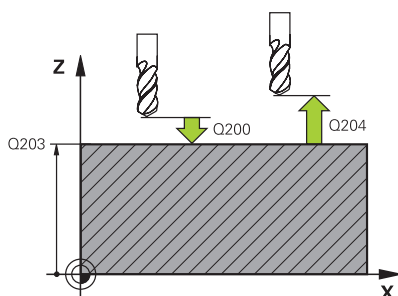
Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**



## Help graphic

## Parameter

**Q301 Move to clearance height (0/1)?**

Specify how the tool moves between machining processes:

**0:** Move to the set-up clearance between operations

**1:** Move to the 2nd set-up clearance between operations

Input: **0, 1**

## Example

11 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=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q301=+1	;MOVE TO CLEARANCE
12 CYCL CALL	

## 8.4 Cycle 224 DATAMATRIX CODE PATTERN (option 19)

ISO programming  
G224

### Application

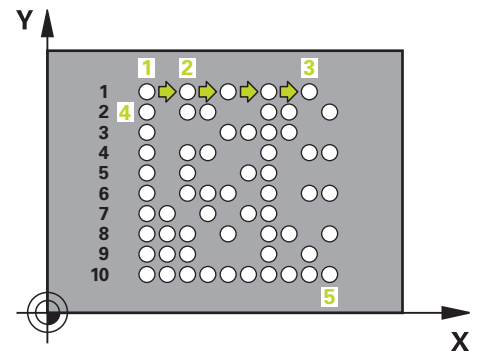


This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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 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 point **2** for the next operation.
- 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 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



## Notes

### NOTICE

#### Danger of collision!

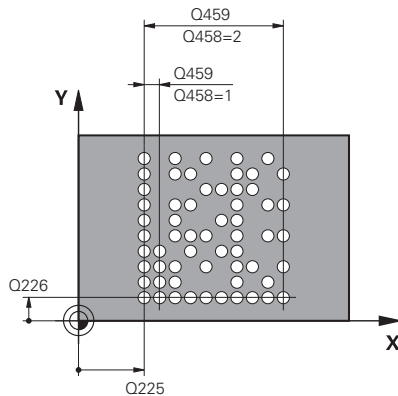
If you combine Cycle **224** with one of the machining cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle **224** will be effective for the selected machining cycle.

- ▶ Check the machining sequence using a graphic simulation
- ▶ Carefully test the NC program or program section in the **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.
- The control uses the special character **%** for special functions. If you want to use this character in a DataMatrix code, enter it twice in the text, e.g. **%%**).

## Cycle parameters

### Help graphic



### Parameter

#### Q225 Starting point in 1st axis?

Coordinate in the lower left corner of the code in the main axis. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q226 Starting point in 2nd axis?

Coordinate in the lower left corner of the code in the secondary axis. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q501 Text input?

Enter the text to be converted within quotation marks. Variables can be assigned.

**Further information:** "Outputting variable texts in DataMatrix codes", Page 254

Input: Max. **255** characters

#### Q458 Cell size/Pattern size(1/2)?

Specify how the DataMatrix code is described in **Q459**:

**1:** Distance between cells

**2:** Pattern size

Input: **1, 2**

#### Q459 Size for pattern?

Definition of the distance between cells or the pattern size:

If **Q458=1**: Distance between the first and second cell (between cell centers)

If **Q458=2**: Distance between the first and last cell (between cell centers)

This value has an incremental effect.

Input: **0...99999.9999**

#### Q224 Angle of rotation?

Angle by which the entire pattern is rotated. The center of rotation lies in the starting point. The value has an absolute effect.

Input: **-360.000...+360.000**

#### Q200 Set-up clearance?

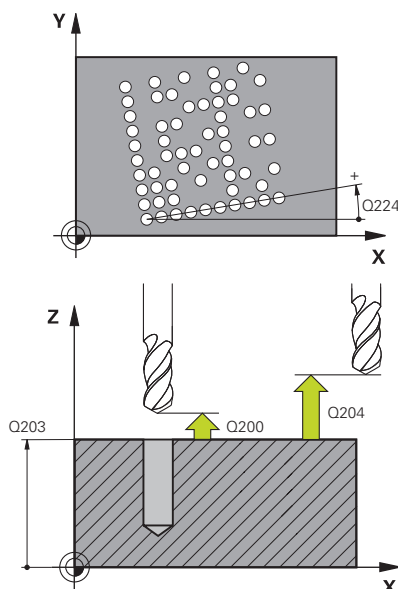
Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**



**Help graphic****Parameter****Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Example**

11 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
12 CYCL CALL	

**Outputting variable texts in DataMatrix codes**

In addition to specified characters you can also output certain variables in DataMatrix codes. Precede the variable with %.

You can use the following variable texts in Cycle **224 DATAMATRIX CODE PATTERN**:

- Date and time
- Names and paths of NC programs
- Count values

### Date and time

You can convert the current date, the current time, or the current calendar week into a DataMatrix code. Enter the value **%time<x>** in cycle parameter **QS501**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY.



Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **%time08**.

The following formats are available:

Input	Format
<b>%time00</b>	DD.MM.YYYY hh:mm:ss
<b>%time01</b>	D.MM.YYYY h:mm:ss
<b>%time02</b>	D.MM.YYYY h:mm
<b>%time03</b>	D.MM.YY h:mm
<b>%time04</b>	YYYY-MM-DD hh:mm:ss
<b>%time05</b>	YYYY-MM-DD hh:mm
<b>%time06</b>	YYYY-MM-DD h:mm
<b>%time07</b>	YY-MM-DD h:mm
<b>%time08</b>	DD.MM.YYYY
<b>%time09</b>	D.MM.YYYY
<b>%time10</b>	D.MM.YY
<b>%time11</b>	YYYY-MM-DD
<b>%time12</b>	YY-MM-DD
<b>%time13</b>	hh:mm:ss
<b>%time14</b>	h:mm:ss
<b>%time15</b>	h:mm
<b>%time99</b>	Calendar week

### Names and paths of NC programs

You can convert the name or path of the active or called NC program into a DataMatrix code. Enter the value **%main<x>** or **%prog<x>** in cycle parameter **QS501**.

The following formats are available:

Input	Meaning	Example
<b>%main0</b>	Full path of the active NC program	<b>TNC:\MILL.h</b>
<b>%main1</b>	Directory path of the active NC program	<b>TNC:\</b>
<b>%main2</b>	Name of the active NC program	<b>MILL</b>
<b>%main3</b>	File type of the active NC program	<b>.H</b>
<b>%prog0</b>	Full path of the called NC program	<b>TNC:\HOUSE.h</b>
<b>%prog1</b>	Directory path of the called NC program	<b>TNC:\</b>
<b>%prog2</b>	Name of the called NC program	<b>HOUSE</b>
<b>%prog3</b>	File type of the called NC program	<b>.H</b>

### Count values

You can convert the current count value into a DataMatrix code. The control displays the current count value in the MOD menu.

Enter the value **%count<x>** in cycle parameter **QS501**.

The number after **%count** indicates how many digits the DataMatrix code contains. The maximum is nine digits.

Example:

- Programming: **%count9**
- Current count value: 3
- Result: 000000003

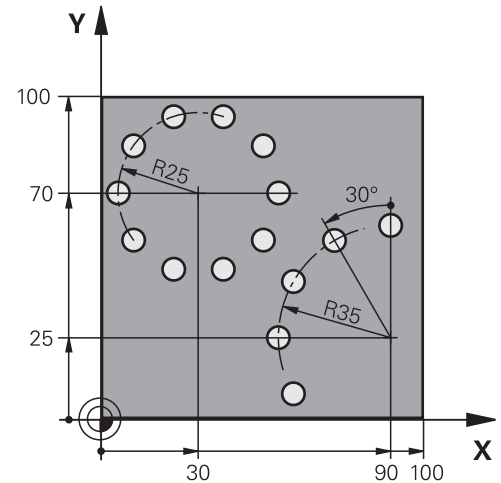
### Operating information

- In the Test Run operating mode, the control only simulates the count value you define directly in the NC program. The count value from the MOD menu is ignored.
- In the SINGLE BLOCK and FULL SEQ. operating modes, the control will take the counter reading from the MOD menu into account.



## 8.5 Programming Examples

### Example: Polar hole patterns



0 BEGIN PGM 200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 200 Z S3500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-15 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+4 ;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	
6 CYCL DEF 220 POLAR PATTERN ~	
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 ~	
	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+100 R0 FMAX	; Retract the tool
9	M30	; End of program
10	END PGM 200 MM	

# 9

**Cycles: Contour  
Pocket**

## 9.1 SL Cycles

### General information

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 have 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, always use a graphic test run to verify your program before running it. 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

- Closed contour without approach and departure movements
- Coordinate transformations are permitted; if they are programmed within the subcontours, 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
- Without machining cycles, feed rates, and M functions

### 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**.





#### Program structure: Machining with SL Cycles

<b>0 BEGIN SL 2 MM</b>
...
<b>12 CYCL DEF 14 CONTOUR</b>
...
<b>13 CYCL DEF 20 CONTOUR DATA</b>
...
<b>16 CYCL DEF 21 PILOT DRILLING</b>
...
<b>17 CYCL CALL</b>
...
<b>22 CYCL DEF 23 FLOOR FINISHING</b>
...
<b>23 CYCL CALL</b>
...
<b>26 CYCL DEF 24 SIDE FINISHING</b>
...
<b>27 CYCL CALL</b>
...
<b>50 L Z+250 R0 FMAX M2</b>
<b>51 LBL 1</b>
...
<b>55 LBL 0</b>
<b>56 LBL 2</b>
...
<b>60 LBL 0</b>
...
<b>99 END PGM SL2 MM</b>

## Overview

Soft key	Cycle	Page
	Cycle 14 CONTOUR ■ Listing the contour subprograms	263
	Cycle 20 CONTOUR DATA (option 19) ■ Input of machining information	267
	Cycle 21 PILOT DRILLING (option 19) ■ Machining a hole for non-center cutting tools	270
	Cycle 22 ROUGH-OUT (option 19) ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	272
	Cycle 23 FLOOR FINISHING (option 19) ■ Finishing with finishing allowance for the floor from Cycle <b>20</b>	277
	Cycle 24 SIDE FINISHING (option 19) ■ Finishing with side finishing allowance from Cycle <b>20</b>	280

### Enhanced cycles:

Soft key	Cycle	Page
	Cycle 270 CONTOUR TRAIN DATA (option 19) ■ Input of contour data for Cycle <b>25</b> or <b>276</b>	283
	Cycle 25 CONTOUR TRAIN (option 19) ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	285
	Cycle 275 TROCHOIDAL SLOT (option 19) ■ Machining of open and closed contours using trochoidal milling.	289
	Cycle 276 THREE-D CONT. TRAIN (option 19) ■ Machining of open and closed contours ■ Detection of residual material ■ 3-D contours—additional processing of coordinates from the tool axis	295

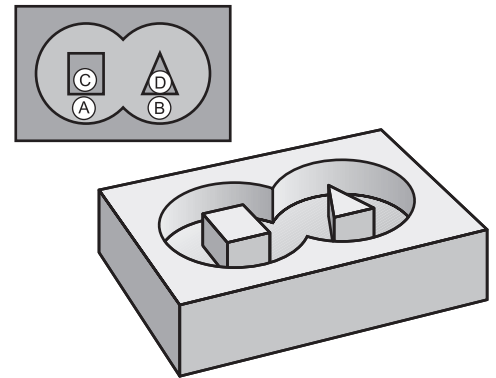
## 9.2 Cycle 14 CONTOUR

### ISO programming

G37

### Application

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



### Notes

- 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

Help graphic	Parameter
	<b>Label numbers for contour?</b> Enter all label numbers for the individual subprograms that are to be superimposed to define a contour. Confirm each number with the ENT key. Confirm your entries with the <b>END</b> key. Up to 12 subprogram numbers are possible. Input: <b>0...65535</b>

### Example

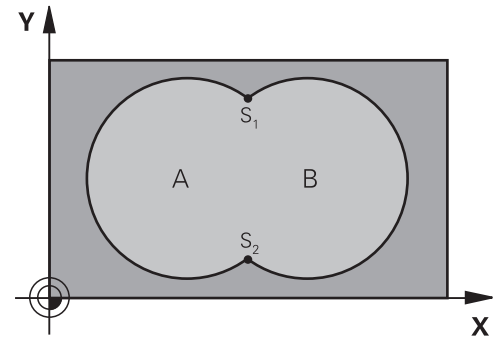
```
11 CYCL DEF 14.0 CONTOUR
```

```
12 CYCL DEF 14.1 CONTOUR LABEL1 /2
```

## 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.



### 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

```
11 LBL 1
12 L X+10 Y+10 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0
```

#### Subprogram 2: Pocket B

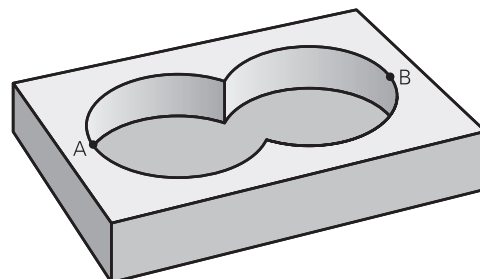
```
16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0
```



### Surface resulting from sum

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:

11 LBL 1
12 L X+10 Y+50 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0

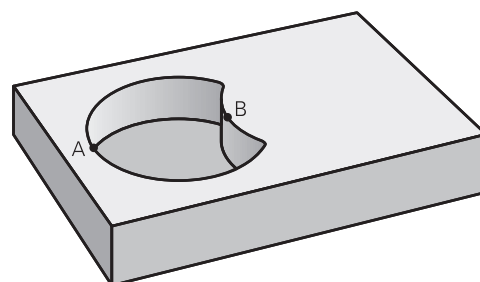
#### Surface B:

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

### Surface resulting from difference

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:

11 LBL 1
12 L X+10 Y+50 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0

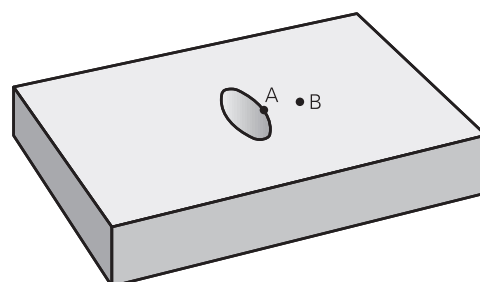
#### Surface B:

16 LBL 2
17 L X+40 Y+50 RL
18 CC X+65 Y+50
19 C X+40 Y+50 DR-
20 LBL 0

### Surface resulting from 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:

11 LBL 1
12 L X+60 Y+50 RR
13 CC X+35 Y+50
14 C X+60 Y+50 DR-
15 LBL 0

#### Surface B:

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

## 9.4 Cycle 20 CONTOUR DATA (option 19)

### ISO programming

G120

### Application



This function must be enabled and adapted by the machine manufacturer.

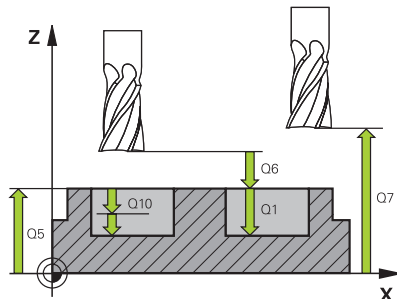
Use Cycle **20** to specify machining data for the subprograms describing the subcontours.

### Notes

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

## Cycle parameters

### Help graphic



### Parameter

#### Q1 Milling depth?

Distance between workpiece surface and pocket floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q2 Path overlap factor?

$Q2 \times \text{tool radius} = \text{stepover factor } k$

Input: **0.0001...1.9999**

#### Q3 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q4 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q5 Workpiece surface coordinate?

Absolute coordinate of the top surface of the workpiece

Input: **-99999.9999...+99999.9999**

#### Q6 Set-up clearance?

Distance between tool tip and the top surface of the workpiece. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q7 Clearance height?

Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q8 Inside corner radius?:

Inside "corner" rounding radius; entered value is referenced to the path of the tool center and is used to calculate smoother traverse motions between the contour elements.

**Q8 is not a radius that is inserted between programmed elements as a separate contour element.**

Input: **0...99999.9999**

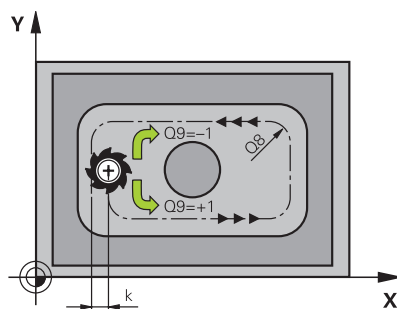
#### Q9 Direction of rotation? cw = -1

Machining direction for pockets

**Q9 = -1** up-cut milling for pocket and island

**Q9 = +1** climb milling for pocket and island

Input: **-1, 0, +1**



**Example**

11 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=+0	;SURFACE COORDINATE ~
Q6=+2	;SET-UP CLEARANCE ~
Q7=+50	;CLEARANCE HEIGHT ~
Q8=+0	;ROUNDING RADIUS ~
Q9=+1	;ROTATIONAL DIRECTION

## 9.5 Cycle 21 PILOT DRILLING (option 19)

### ISO programming

G121

### Application



This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 1 The control 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 behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

## Notes

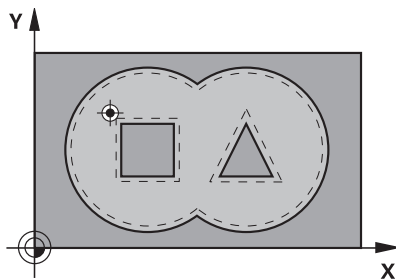
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.
- In narrow areas, the control may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.
- If **Q13=0**, the control uses the data of the tool that is currently in the spindle.

### Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining. After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have programmed **ToolAxClearanceHeight**.

## Cycle parameters

### Help graphic



### Parameter

#### Q10 Plunging depth?

Tool infeed per cut (minus sign for negative machining direction). This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q11 Feed rate for plunging?

Tool traversing speed in mm/min during plunging

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q13 or QS13 Rough-out tool number/name?

Number or name of the rough-out tool. You are able to transfer the tool directly from the tool table via soft key.

Input: **0...999999.9** or max. **255** characters

### Example

11 CYCL DEF 21 PILOT DRILLING ~	
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q13=+0	;ROUGH-OUT TOOL

## 9.6 Cycle 22 ROUGH-OUT (option 19)

### ISO programming

G122

### Application



This function must be enabled and adapted by the machine manufacturer.

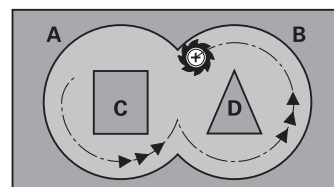
Use Cycle **22 ROUGH-OUT** to define the technology data for roughing.

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 sequence

- 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 behavior depends on the machine parameter **posAfterContPocket** (no. 201007).





## Notes

### 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; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- 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 for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q1**, the control will display an error message.



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

## Notes on programming

- 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.
- 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** is defined 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**))

**Note regarding machine parameters**

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

## Cycle parameters

Help graphic	Parameter
	<b>Q10 Plunging depth?</b> Tool infeed per cut. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q11 Feed rate for plunging?</b> Traversing feed rate in the spindle axis Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q12 Feed rate for roughing?</b> Traversing feed rate in the working plane Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q18 or QS18 Coarse roughing tool?</b> Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the <b>Tool name</b> 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 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 <b>LCUTS</b> in the TOOL.T tool table and define the maximum plunging angle of the tool with <b>ANGLE</b> . Input: <b>0...99999.9</b> or max. <b>255</b> characters
	<b>Q19 Feed rate for reciprocation?</b> Reciprocation feed rate in mm/min Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q208 Feed rate for retraction?</b> Tool traversing speed in mm/min when retracting after the machining operation. If you enter <b>Q208 = 0</b> , the control retracts the tool at the feed rate specified in <b>Q12</b> . Input: <b>0...99999.9999</b> or <b>FMAX, FAUTO, PREDEF</b>

## Help graphic

## Parameter

**Q401 Feed rate factor in %?**

Percentage value to 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: **0.0001...100**

**Q404 Fine roughing strategy (0/1)?**

Define how the control will move the tool during fine roughing when the radius of the fine-roughing tool is equal to or larger than half the radius of the coarse roughing tool.

**0:** Between areas that need to be fine-roughed, the control moves the tool along the contour at the current depth

**1:** Between areas that need to be fine-roughed, the control retracts the tool to set-up clearance and then moves it to the starting point of the next area to be roughed out

Input: **0, 1**

## Example

11 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q18=+0	;COARSE ROUGHING TOOL ~
Q19=+0	;FEED RATE FOR RECIP. ~
Q208=+99999	;RETRACTION FEED RATE ~
Q401=+100	;FEED RATE FACTOR ~
Q404=+0	;FINE ROUGH STRATEGY

## 9.7 Cycle 23 FLOOR FINISHING (option 19)

### ISO programming

G123

### Application



This function must be enabled and adapted by the machine manufacturer.

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
- Cycle **22 ROUGH-OUT**, if necessary

### Cycle sequence

- 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 behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

## Notes

### 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; do not program an incremental traversing movement

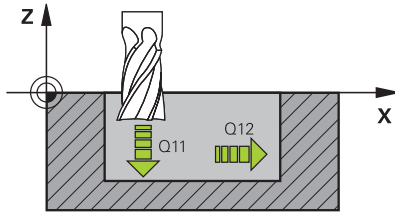
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.
- The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside 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.

#### Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

## Cycle parameters

### Help graphic



### Parameter

#### Q11 Feed rate for plunging?

Tool traversing speed in mm/min during plunging

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q12 Feed rate for roughing?

Traversing feed rate in the working plane

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q208 Feed rate for retraction?

Tool traversing speed 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: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

### Example

11 CYCL DEF 23 FLOOR FINISHING ~

Q11=+150 ;FEED RATE FOR PLNGNG ~

Q12=+500 ;FEED RATE F. ROUGHNG ~

Q208=+99999 ;RETRACTION FEED RATE

## 9.8 Cycle 24 SIDE FINISHING (option 19)

### ISO programming

G124

### Application



This function must be enabled and adapted by the machine manufacturer.

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

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
- Cycle **22 ROUGH-OUT**, if necessary

### Cycle sequence

- 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 behavior depends on the machine parameter **posAfterContPocket** (no. 201007).



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.



## Notes

### 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; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If no allowance was defined in Cycle **20**, the control generates the error message "Tool radius too large."
- If you run Cycle **24** without having roughed out with Cycle **22**, then enter "0" for the radius of the rough mill.
- 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 for arcs compensated on the inside 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.

#### Notes on programming

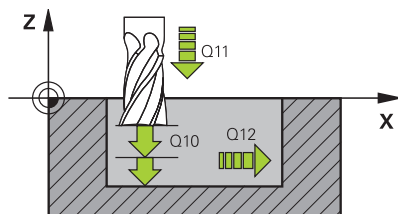
- The sum of finishing allowance for the side (**Q14**) and the radius of the finish mill must be smaller than the sum of allowance for side (**Q3**, Cycle **20**) and the radius of the rough mill.
- The finishing allowance for the side **Q14** is left over after finishing. Therefore, it must be smaller than the allowance in Cycle **20**.
- 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 boundary)
  - In Cycle **20**, enter a finishing allowance (**Q3**) greater than the sum of the finishing allowance **Q14** + radius of the tool being used

#### Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket:
  - **PosBeforeMachining**: Return to starting position.
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

## Cycle parameters

### Help graphic



### Parameter

#### Q9 Direction of rotation? cw = -1

Machining direction:

**+1**: Counterclockwise

**-1**: Clockwise

Input: **-1, +1**

#### Q10 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q11 Feed rate for plunging?

Tool traversing speed in mm/min during plunging

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q12 Feed rate for roughing?

Traversing feed rate in the working plane

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q14 Finishing allowance for side?

The finishing allowance for the side **Q14** is left over after finishing.

This allowance must be smaller than the allowance in Cycle **20**.

This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q438 or QS438 Number/name of rough-out tool?

Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name 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 is 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: **-1...+32767.9** or **255** characters

### Example

11 CYCL DEF 24 SIDE FINISHING ~	
Q9=+1	;ROTATIONAL DIRECTION ~
Q10=+5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q14=+0	;ALLOWANCE FOR SIDE ~
Q438=-1	;ROUGH-OUT TOOL

## 9.9 Cycle 270 CONTOUR TRAIN DATA (option 19)

ISO programming  
G270

### Application



This function must be enabled and adapted by the machine manufacturer.

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

### Notes

- 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

Help graphic	Parameters
	<b>Q390 Type of approach/departure?</b> Definition of type of approach/departure: <b>1:</b> Approach the contour tangentially on a circular arc <b>2:</b> Approach the contour tangentially on a straight line <b>3:</b> Approach the contour at a right angle <b>0 and 4:</b> No approach or departure movement is performed. Input: <b>1, 2, 3</b>
	<b>Q391 Radius comp. (0=R0/1=RL/2=RR)?</b> Definition of radius compensation: <b>0:</b> Machine the defined contour without radius compensation <b>1:</b> Machine the defined contour with compensation to the left <b>2:</b> Machine the defined contour with compensation to the right Input: <b>0, 1, 2</b>
	<b>Q392 App. radius/dep. radius?</b> Only in effect if a tangential approach on a circular path was selected ( <b>Q390</b> = 1). Radius of the approach/departure arc Input: <b>0...99999.9999</b>
	<b>Q393 Center angle?</b> Only in effect if a tangential approach on a circular path was selected ( <b>Q390</b> = 1). Angular length of the approach arc Input: <b>0...99999.9999</b>
	<b>Q394 Distance from aux. point?</b> Only in effect if a tangential approach on a straight line or a right-angle approach is selected ( <b>Q390</b> = 2 or <b>Q390</b> = 3). Distance to the auxiliary point from which the tool will approach the contour. Input: <b>0...99999.9999</b>

### Example

11 CYCL DEF 270 CONTOUR TRAIN DATA ~	
Q390=+1	;TYPE OF APPROACH ~
Q391=+1	;RADIUS COMPENSATION ~
Q392=+5	;RADIUS ~
Q393=+90	;CENTER ANGLE ~
Q394=+0	;DISTANCE

## 9.10 Cycle 25 CONTOUR TRAIN (option 19)

### ISO programming

G125

### Application

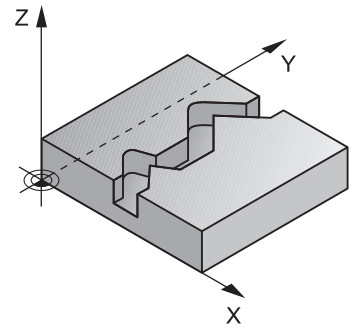


This function must be enabled and adapted by the machine manufacturer.

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.



## Notes

### 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; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control takes only the first label of Cycle **14 CONTOUR** into account.
- 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 for arcs compensated on the inside will be reduced accordingly.

#### Notes on programming

- Cycle **20 CONTOUR DATA**, is not required.
- 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 use local **QL Q** parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

## Cycle parameters

Help graphic	Parameter
	<b>Q1 Milling depth?</b> Distance between workpiece surface and contour floor. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q3 Finishing allowance for side?</b> Finishing allowance in the working plane. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q5 Workpiece surface coordinate?</b> Absolute coordinate of the top surface of the workpiece Input: <b>-99999.9999...+99999.9999</b>
	<b>Q7 Clearance height?</b> Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q10 Plunging depth?</b> Tool infeed per cut. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q11 Feed rate for plunging?</b> Traversing feed rate in the spindle axis Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q12 Feed rate for roughing?</b> Traversing feed rate in the working plane Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q15 Climb or up-cut? up-cut = -1</b> <b>+1:</b> Climb milling <b>-1:</b> Up-cut milling <b>0:</b> Climb milling and up-cut milling alternately in several infeeds Input: <b>-1, 0, +1</b>

## Help graphic

## Parameter

**Q18 or QS18 Coarse roughing tool?**

Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name 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 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**.

Input: **0...99999.9** or max. **255** characters

**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: **0.001...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: **0...999.999**

**Q448 Path extension?**

Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour.

Input: **0...99.999**

## Example

11 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=+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



## 9.11 Cycle 275 TROCHOIDAL SLOT (option 19)

### ISO programming

G275

### Application



This function must be enabled and adapted by the machine manufacturer.

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 CYC275 MM

...

12 CYCL DEF 14 CONTOUR

...

13 CYCL DEF 275 TROCHOIDAL SLOT

...

14 CYCL CALL M3

...

50 L Z+250 R0 FMAX M2

51 LBL 10

...

55 LBL 0

...

99 END PGM CYC275 MM

**Cycle sequence****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 plunges vertically 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

## Notes

### 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; do not program an incremental traversing movement

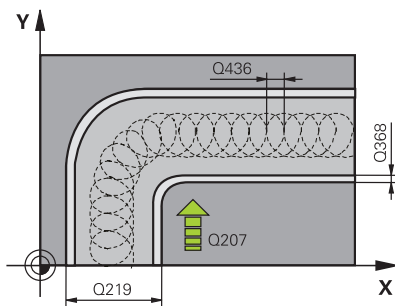
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- In conjunction with Cycle **275**, the control does not require Cycle **20 CONTOUR DATA**.

#### Notes on programming

- 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 starting point of a closed slot must not be located in a contour corner.

## Cycle parameters

### Help graphic



### Parameter

#### Q215 Machining operation (0/1/2)?

Define the machining operation:

**0:** Roughing and finishing

**1:** Only roughing

**2:** Only finishing

Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined

Input: **0, 1, 2**

#### Q219 Width of slot?

Enter the width of the slot. It is in parallel to the secondary axis of the working plane. If you enter a slot width that equals the tool diameter, then the control will carry out the roughing process only (slot milling).

Maximum slot width for roughing: Twice the tool diameter

Input: **0...99999.9999**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q436 Feed per revolution?

Value by which the control moves the tool in the machining direction per revolution. The value has an absolute effect.

Input: **0...99999.9999**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

#### Q351 Direction? Climb=+1, Up-cut=-1

Type of milling operation. The direction of spindle rotation is taken into account.

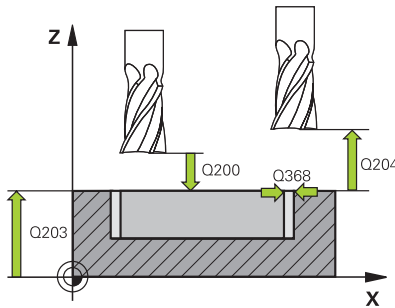
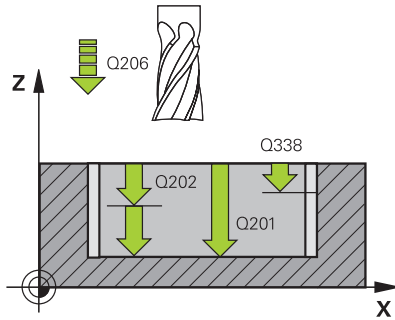
**+1** = climb milling

**-1** = up-cut milling

**PREDEF:** The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

## Help graphic



## Parameter

**Q201 Depth?**

Distance between workpiece surface and slot floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

**Q202 Plunging depth?**

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

**Q206 Feed rate for plunging?**

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q338 Infeed for finishing?**

Tool infeed in the spindle axis per finishing cut.

**Q338 = 0:** Finishing with a single infeed

This value has an incremental effect.

Input: **0...99999.9999**

**Q385 Finishing feed rate?**

Traversing speed of the tool in mm/min for side and floor finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q366 Plunging strategy (0/1/2)?**

Type of plunging strategy:

**0** = Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle ANGLE defined in the tool table

**1** = No function

**2** = Reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message

Input: **0, 1, 2** or **PREDEF**

## Help graphic

## Parameter

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q439 Feed rate reference (0-3)?**

Specify the reference for the programmed feed rate:

**0:** Feed rate is referenced to the path of the tool center

**1:** Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center

**2:** Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center

**3:** Feed rate is always referenced to the cutting edge

Input: **0, 1, 2, 3**

## Example

11 CYCL DEF 275 TROCHOIDAL SLOT ~	
Q215=+0	;MACHINING OPERATION ~
Q219=+10	;SLOT WIDTH ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q436=+2	;INFED 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=+0	;INFED 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
12 CYCL CALL	

## 9.12 Cycle 276 THREE-D CONT. TRAIN (option 19)

ISO programming  
G276

### Application

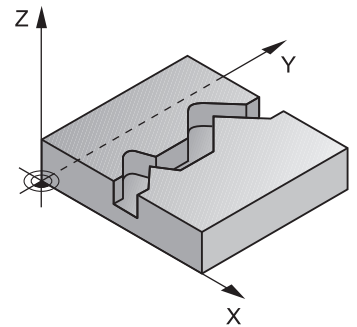


This function must be enabled and adapted by the machine manufacturer.

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 sequence****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 is 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



## Notes

### 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; do not program an incremental traversing movement

### 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.
- 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.
- 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.
- 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 for arcs compensated on the inside will be reduced accordingly.

## Notes on programming

- The first NC block in the contour subprogram must contain values in all of the three axes X, Y and Z.
- The algebraic sign for the depth parameter determines the working direction. If you program **DEPTH = 0**, the control will use the tool axis coordinates that have been specified in the contour subprogram.
- If you use local **QL Q** parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

## Cycle parameters

Help graphic	Parameter
	<b>Q1 Milling depth?</b> Distance between workpiece surface and contour floor. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q3 Finishing allowance for side?</b> Finishing allowance in the working plane. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q7 Clearance height?</b> Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q10 Plunging depth?</b> Tool infeed per cut. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q11 Feed rate for plunging?</b> Traversing feed rate in the spindle axis Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q12 Feed rate for roughing?</b> Traversing feed rate in the working plane Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q15 Climb or up-cut? up-cut = -1</b> <b>+1:</b> Climb milling <b>-1:</b> Up-cut milling <b>0:</b> Climb milling and up-cut milling alternately in several infeeds Input: <b>-1, 0, +1</b>
	<b>Q18 or QS18 Coarse roughing tool?</b> Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the <b>Tool name</b> 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 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 <b>LCUTS</b> in the TOOL.T tool table and define the maximum plunging angle of the tool with <b>ANGLE</b> . Input: <b>0...99999.9</b> or max. <b>255</b> characters

## Help graphic

## Parameter

**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: **0.001...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: **0...999.999**

**Q448 Path extension?**

Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour.

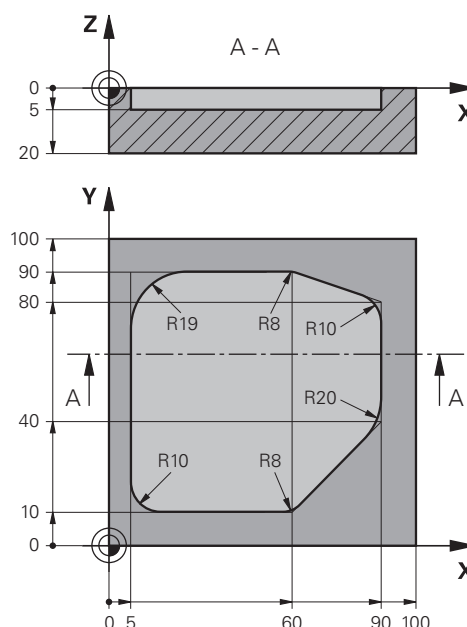
Input: **0...99.999**

## Example

11 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

## 9.13 Programming examples

### Example: Roughing-out and fine-roughing a pocket with SL Cycles



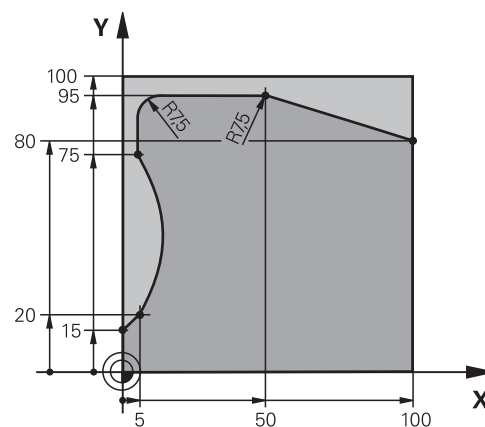
0 BEGIN PGM 1078634 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 15 Z S4500	; Tool call: coarse roughing tool (diameter: 30)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-5 ;MILLING 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=+50 ;CLEARANCE HEIGHT ~	
Q8=+0.2 ;ROUNDING RADIUS ~	
Q9=+1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	

Q208=+99999	;RETRACTION FEED RATE ~	
Q401=+90	;FEED RATE FACTOR ~	
Q404=+1	;FINE ROUGH STRATEGY	
9 CYCL CALL		; Cycle call: coarse roughing
10 L Z+200 R0 FMAX		; Retract the tool
11 TOOL CALL 4 Z S3000		; Tool call: fine roughing tool (diameter: 8)
12 L Z+100 R0 FMAX M3		
13 CYCL DEF 22 ROUGH-OUT ~		
Q10=-5	;PLUNGING DEPTH ~	
Q11=+150	;FEED RATE FOR PLNGNG ~	
Q12=+500	;FEED RATE F. ROUGHNG ~	
Q18=+15	;COARSE ROUGHING TOOL ~	
Q19=+200	;FEED RATE FOR RECIP. ~	
Q208=+99999	;RETRACTION FEED RATE ~	
Q401=+90	;FEED RATE FACTOR ~	
Q404=+1	;FINE ROUGH STRATEGY	
14 CYCL CALL		; Cycle call: fine roughing
15 L Z+200 R0 FMAX		; Retract the tool
16 M30		; End of program
17 LBL 1		; Contour subprogram
18 L X+5 Y+50 RR		
19 L Y+90		
20 RND R19		
21 L X+60		
22 RND R8		
23 L X+90 Y+80		
24 RND R10		
25 L Y+40		
26 RND R20		
27 L X+60 Y+10		
28 RND R8		
29 L X+5		
30 RND R10		
31 L X+5 Y+50		
32 LBL 0		
33 END PGM 1078634 MM		

302

Q208=+99999	;RETRACTION FEED RATE ~	
Q401=+100	;FEED RATE FACTOR ~	
Q404=+0	;FINE ROUGH STRATEGY	
13 CYCL CALL		; Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING ~		
Q11=+100	;FEED RATE FOR PLNGNG ~	
Q12=+200	;FEED RATE F. ROUGHNG ~	
Q208=+99999	;RETRACTION FEED RATE	
15 CYCL CALL		; Cycle call: floor finishing
16 CYCL DEF 24 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 ~	
Q438=-1	;ROUGH-OUT TOOL	
17 CYCL CALL		; Cycle call: side finishing
18 L Z+100 R0 FMAX		; Retract the tool
19 M30		; End of program
20 LBL 1		; Contour subprogram 1: left pocket
21 CC X+35 Y+50		
22 L X+10 Y+50 RR		
23 C X+10 DR-		
24 LBL 0		
25 LBL 2		; Contour subprogram 2: right pocket
26 CC X+65 Y+50		
27 L X+90 Y+50 RR		
28 C X+90 DR-		
29 LBL 0		
30 LBL 3		; Contour subprogram 3: left square island
31 L X+27 Y+50 RL		
32 L Y+58		
33 L X+43		
34 L Y+42		
35 L X+27		
36 LBL 0		
37 LBL 4		; Contour subprogram 4: right triangular island
38 L X+65 Y+42 RL		
39 L X+57		
40 L X+65 Y+58		
41 L X+73 Y+42		
42 LBL 0		
43 END PGM 2 MM		

### Example: Contour train



0 BEGIN PGM 3 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S2000	; Tool call (diameter: 20)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 25 CONTOUR TRAIN ~	
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 ~
Q18=+0	;COARSE ROUGHING TOOL ~
Q446=+0.01	;RESIDUAL MATERIAL ~
Q447=+10	;CONNECTION DISTANCE ~
Q448=+2	;PATH EXTENSION
8 CYCL CALL	; Cycle call
9 L Z+250 R0 FMAX	; Retract the tool, end program
10 M30	
11 LBL 1	; Contour subprogram
12 L X+0 Y+15 RL	
13 L X+5 Y+20	
13 CT X+5 Y+75	
14 CT X+5 Y+75	
15 L Y+95	
16 RND R7.5	
17 L X+50	



18 RND R7.5	
19 L X+100 Y+80	
20 LBL 0	
21 END PGM 3 MM	



# 10

**Cycles: Optimized  
Contour Milling**

## 10.1 OCM cycles (option 167)

### OCM cycles

#### 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, always perform a graphic test run ! 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** figure 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 figure cycles:**

The figure defined in an OCM figure 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 figure using cycles **127x**
- ▶ If the first figure 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 minimum 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 OCM MM
...
12 CONTOUR DEF
...
13 CYCL DEF 271 OCM CONTOUR DATA
...
16 CYCL DEF 272 OCM ROUGHING
...
17 CYCL CALL
...
20 CYCL DEF 273 OCM FINISHING FLOOR
...
21 CYCL CALL
...
24 CYCL DEF 274 OCM FINISHING SIDE
...
25 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM OCM MM







```

## Overview

### OCM cycles:

Soft key	Cycle	Page
	Cycle 271 OCM CONTOUR DATA (option 167) <ul style="list-style-type: none"> <li>■ Definition of the machining information for the contour or subprograms</li> <li>■ Input of a bounding frame or block</li> </ul>	312
	Cycle 272 OCM ROUGHING (option 167) <ul style="list-style-type: none"> <li>■ Technology data for roughing contours</li> <li>■ Use of the OCM cutting data calculator</li> <li>■ Plunging behavior: vertical, helical, or reciprocating</li> <li>■ Plunging strategy: selectable</li> </ul>	315
	Cycle 273 OCM FINISHING FLOOR (option 167) <ul style="list-style-type: none"> <li>■ Finishing with finishing allowance for the floor from Cycle <b>271</b></li> <li>■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances)</li> </ul>	330
	Cycle 274 OCM FINISHING SIDE (option 167) <ul style="list-style-type: none"> <li>■ Finishing with side finishing allowance from Cycle <b>271</b></li> </ul>	334
	Cycle 277 OCM CHAMFERING (option 167) <ul style="list-style-type: none"> <li>■ Deburr the edges</li> <li>■ Consider adjacent contours and walls</li> </ul>	337

### OCM standard figures:

Soft key	Cycle	Page
	Cycle 1271 OCM RECTANGLE (option 167) <ul style="list-style-type: none"> <li>■ Definition of a rectangle</li> <li>■ Input of the side lengths</li> <li>■ Definition of the corners</li> </ul>	343
	Cycle 1272 OCM CIRCLE (option 167) <ul style="list-style-type: none"> <li>■ Definition of a circle</li> <li>■ Input of the circle diameter</li> </ul>	346
	Cycle 1273 OCM SLOT / RIDGE (option 167) <ul style="list-style-type: none"> <li>■ Definition of a groove or ridge</li> <li>■ Input of the width and the length</li> </ul>	349
	Cycle 1278 OCM POLYGON (option 167) <ul style="list-style-type: none"> <li>■ Definition of a polygon</li> <li>■ Input of the reference circle</li> <li>■ Definition of the corners</li> </ul>	352
	Cycle 1281 OCM RECTANGLE BOUNDARY (option 167) <ul style="list-style-type: none"> <li>■ Definition of a bounding rectangle</li> </ul>	355
	Cycle 1282 OCM CIRCLE BOUNDARY (option 167) <ul style="list-style-type: none"> <li>■ Definition of a bounding circle</li> </ul>	357

## 10.2 Cycle 271 OCM CONTOUR DATA (option 167)

ISO programming  
G271

### 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.

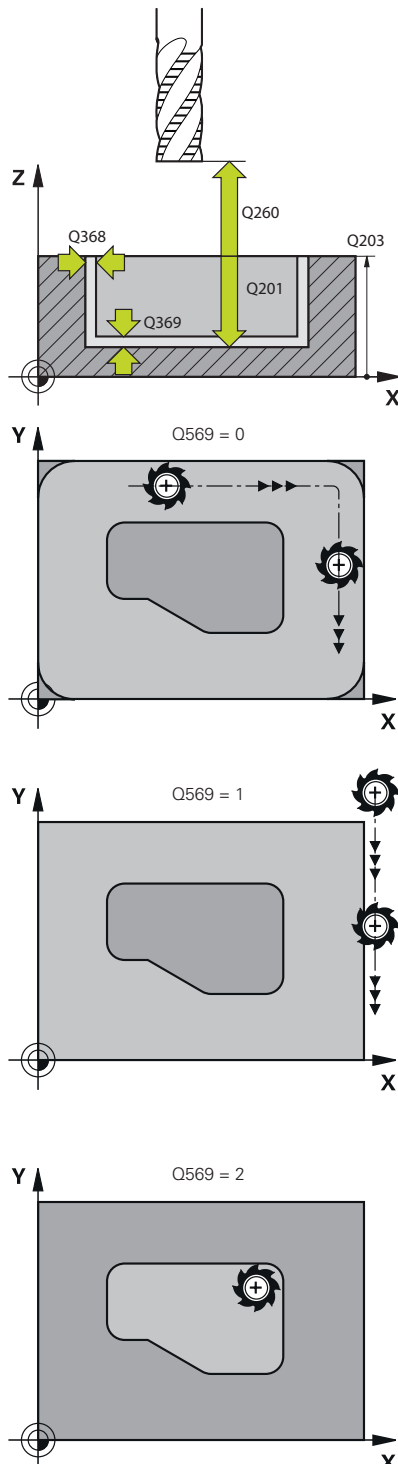
### Notes

- 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

### Help graphic



### Parameter

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q201 Depth?

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q369 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

#### Q260 Clearance height?

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

#### 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: **0.05...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

Input: **0, 1, 2**

**Example**

11 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 Cycle 272 OCM ROUGHING (option 167)

### ISO programming

#### G272

### 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 321

### Requirements

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

- **CONTOUR DEF / SEL CONTOUR** or Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**

### Cycle sequence

- 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
- 3 The control moves to the first plunging depth. The plunging depth and the sequence for machining the contours depend on the plunging strategy **Q575**.

Depending on the definition in Cycle **271 OCM CONTOUR DATA**, parameter **Q569 OPEN BOUNDARY**, the control plunges as follows:

- **Q569 = 0 or 2:** The tool plunges into the material in a helical or reciprocating movement. The finishing allowance for the side is taken into account.

**Further information:** "Plunging behavior with Q569 = 0 or 2", Page 316

- **Q569 = 1:** The tool plunges vertically outside the open boundary to the first plunging depth
- 4 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**
  - 5 In the next step, the tool is moved to the next plunging depth and repeats the roughing procedure until the programmed contour is completely machined
  - 6 Finally, the tool retracts in the tool axis to the clearance height
  - 7 If there are more contours, the control will repeat the machining process. The control then moves to the contour whose starting point is positioned nearest to the current tool position (depending on the infeed strategy **Q575**).

**Plunging behavior with Q569 = 0 or 2**

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:

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

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:

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

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

## Notes

### NOTICE

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

The cycle does not include the corner radius **R2** in the calculation of the milling paths. Even if you use a small overlap factor, residual material may be left over on the contour floor. The residual material can cause damage to the workpiece and the tool during subsequent machining operations!

- ▶ Run a simulation to verify the machining sequence and the contour
- ▶ Use tools without a corner radius **R2** where possible

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.
- 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.



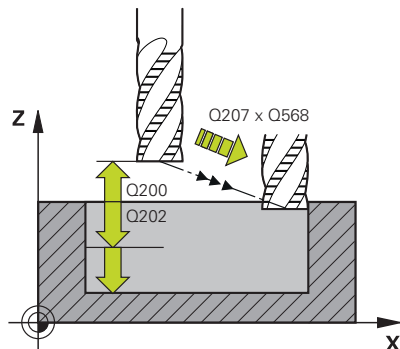
If required, use a center-cut end mill (ISO 1641).

#### Notes on programming

- **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 path overlap factor **Q370 < 1**, a value of less than 1 is also recommended for the plunging factor **Q579**.

## Cycle parameters

### Help graphic



### Parameter

#### Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

Input: **0...99999.9999**

#### Q370 Path overlap factor?

**Q370** x tool radius = lateral infeed *k* on a straight line. The control maintains this value as precisely as possible.

Input: **0.04...1.99** or **PREDEF**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **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: **0.1...1**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q200 Set-up clearance?

Distance between lower edge of tool and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q438 or QS438 Number/name of rough-out tool?

Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field.

**-1**: The control assumes that the tool last used in Cycle **272** is the rough-out tool (default behavior)

**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: **-1...+32767.9** or max. **255** characters

## Help graphic

## Parameter

**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: **0.15...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 of a **GLOBAL DEF** block  
 (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Q576 Spindle speed?**

Spindle speed in revolutions per minute (rpm) for the roughing tool.

**0**: The spindle speed from the **TOOL CALL** block will be used

**> 0**: If a value greater than zero is entered, then this spindle speed will be used

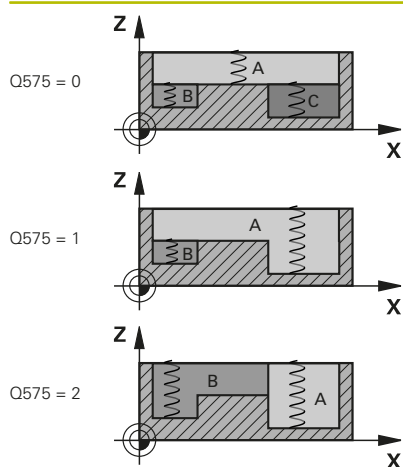
Input: **0...99999**

**Q579 Factor for plunging speed?**

Factor by which the control reduces the **SPINDLE SPEED Q576** for downfeed into the material.

Input: **0.2...1.5**

## Help graphic



## Parameter

**Q575 Infeed strategy (0/1)?**

Type of downfeed:

**0:** The control machines the contour from top to bottom

**1:** The control machines the contour from bottom to top. The control does not always start with the deepest contour. The machining sequence is automatically calculated by the control. The total plunging path is often shorter than with strategy **2**.

**2:** The control machines the contour from bottom to top. The control does not always start with the deepest contour. This strategy calculates the machining sequence such that the maximum length of the cutting edge is used. The resulting total plunging path is thus often larger than with strategy **1**. Depending on **Q568**, this may also result in a shorter machining time.

Input: **0, 1, 2**



The total plunging path is the sum of all plunging movements.

## Example

11 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



## 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.

#### Requirements



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

To open the cutting data calculator:



- Edit Cycle **272 OCM ROUGHING**



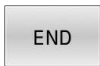
- Press the **OCM CUTTING DATA** soft key
- The control opens the OCM cutting data calculator form.

### Closing the cutting data calculator

To close the cutting data calculator:



- 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

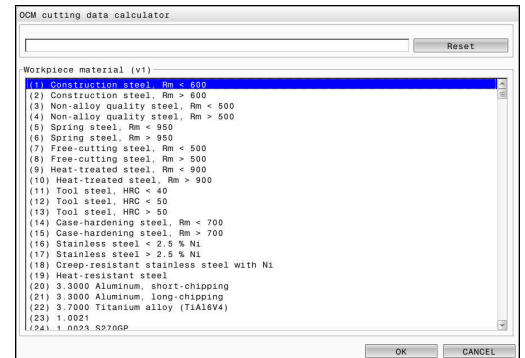


The input fields of the workpiece material and the tool are gray. You can change them only through the selection list or the tool table.

### Workpiece material

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



#### Programming and operating notes:

- 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.

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 Value is applied automatically after the roughing tool has been selected. Input: <b>1...40</b>
Number of teeth	Number of teeth of the roughing tool Value is applied automatically after the roughing tool has been selected. Input: <b>1...10</b>
Angle of twist	Angle of twist of the roughing tool in ° If there are different angles of twist, then enter the average value. Input: <b>0...80</b>



Programming and operating notes:

- 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 feed rate. 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: Input: <b>1...99999</b>
Max. milling speed	Maximum milling speed (feed rate) in mm/min permitted by the machine and the clamping situation: Input: <b>1...99999</b>

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)	<p>Plunging depth (&gt;0 mm to [6 times the tool diameter])</p> <p>The value from cycle parameter <b>Q202</b> is applied when starting the OCM cutting data calculator.</p> <p>Input: <b>0.001...99999.999</b>,</p>
Mechanical load on tool	<p>Slider for selection of the mechanical load (the value is normally between 70 % and 100 %)</p> <p>Input: <b>0%...150%</b></p>
Thermal load on tool	<p>Slider for selection of the thermal load</p> <p>Set the slider according to the thermal wear-resistance (coating) of your tool.</p> <ul style="list-style-type: none"><li>■ HSS: low thermal wear-resistance</li><li>■ VHM (uncoated or normally-coated solid carbide milling cutters): medium thermal wear-resistance</li><li>■ Coated (fully-coated solid carbide milling cutters): high thermal wear-resistance</li></ul> <div><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.</p><p>■ If the slider is in the red range, the control will use the maximum permissible value.</p></div> <p>Input: <b>0%...200%</b></p>

Further information: "Process parameters ", Page 327

### 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)	<b>Q370 = TOOL PATH OVERLAP</b>
Milling feed(Q207) in mm/min	<b>Q207 = FEED RATE MILLING</b>
Spindle speed(Q576) in rpm	<b>Q576 = SPINDLE SPEED</b>
Climb or up-cut(Q351)	<b>Q351= CLIMB OR UP-CUT</b>



Programming and operating notes:

- 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 OCM cutting data calculator compares the cutting data with the input ranges of the cycle. If the values fall below or exceed the input ranges, the parameter will be highlighted in red in the OCM cutting data calculator. In this case, the cutting data cannot be transferred to the cycle.

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 cm<sup>3</sup>/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 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"> <li>■ Reduce the plunging depth</li> <li>■ Use a tool with a lower angle of twist</li> </ul>
Excessive radial force on spindle bearing	Decrease	-	

## 10.5 Cycle 273 OCM FINISHING FLOOR (option 167)

ISO programming  
G273

### 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 sequence

- 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

### Notes

#### NOTICE

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

The cycle does not include the corner radius **R2** in the calculation of the milling paths. Even if you use a small overlap factor, residual material may be left over on the contour floor. The residual material can cause damage to the workpiece and the tool during subsequent machining operations!

- ▶ Run a simulation to verify the machining sequence and the contour
- ▶ Use tools without a corner radius **R2** where possible

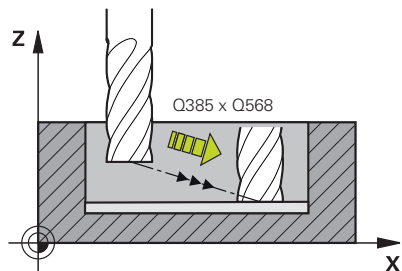
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.
- For finishing with Cycle **273**, the tool always works in climb milling mode.
- 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.

**Note on programming**

- 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

### Help graphic



### Parameter

#### Q370 Path overlap factor?

**Q370** x tool radius = lateral infeed k. The overlap is considered to be the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners.

Input: **0.0001...1.9999** or **PREDEF**

#### Q385 Finishing feed rate?

Traversing speed of the tool in mm/min for floor finishing

Input: **0...99999.999** or **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: **0.1...1**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q200 Set-up clearance?

Distance between lower edge of tool and workpiece surface. This value has an incremental effect.

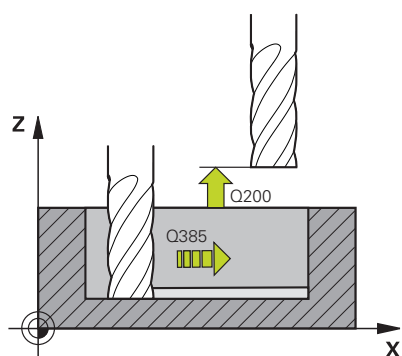
Input: **0...99999.9999** or **PREDEF**

#### Q438 or QS438 Number/name of rough-out tool?

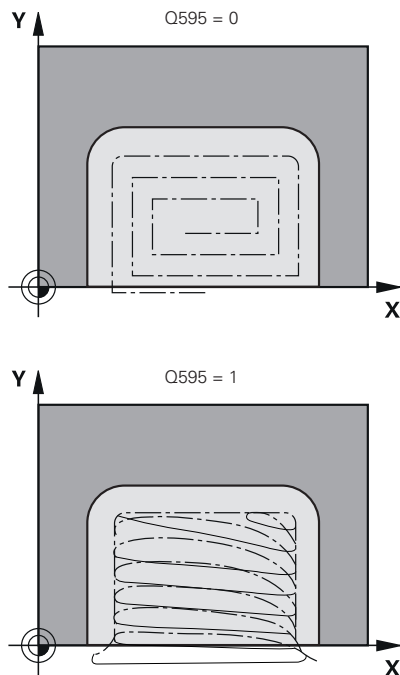
Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field.

**-1:** The control assumes that the tool last used is the rough-out tool (default behavior).

Input: **-1...+32767.9** or max. **255** characters



## Help graphic



## Parameter

**Q595 Strategy (0/1)?**

Machining strategy for finishing

**0:** Equidistant strategy = constant distances between paths**1:** Strategy with constant contact angleInput: **0, 1****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: **0.15...0.99**

## Example

11 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 Cycle 274 OCM FINISHING SIDE (option 167)

### ISO programming G274

#### 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.

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

#### 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 sequence

- 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.

#### Notes

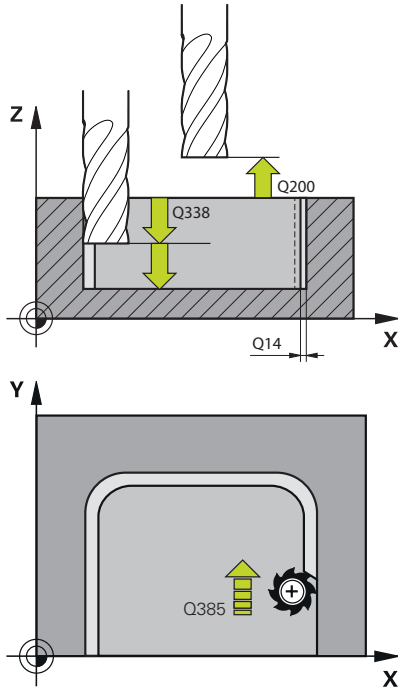
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour and the allowance programmed in Cycle **271**.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

#### Note on programming

- The finishing allowance for the side **Q14** is left over after finishing. It must be smaller than the allowance in Cycle **271**.

## Cycle parameters

### Help graphic



### Parameter

#### Q338 Infeed for finishing?

Tool infeed in the spindle axis per finishing cut.

**Q338 = 0:** Finishing with a single infeed

This value has an incremental effect.

Input: **0...99999.9999**

#### Q385 Finishing feed rate?

Traversing speed of the tool in mm/min for side finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q200 Set-up clearance?

Distance between lower edge of tool and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q14 Finishing allowance for side?

The finishing allowance for the side **Q14** is left over after finishing.

This allowance must be smaller than the allowance in Cycle **271**.

This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q438 or QS438 Number/name of rough-out tool?

Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field.

**-1:** The control assumes that the tool last used is the rough-out tool (default behavior).

Input: **-1...+32767.9** or max. **255** characters

#### 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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

**Example**

11 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0	;INFED FOR FINISHING ~
Q385=+500	;FINISHING FEED RATE ~
Q253=+750	;F PRE-POSITIONING ~
Q200=+2	;SET-UP CLEARANCE ~
Q14=+0	;ALLOWANCE FOR SIDE ~
Q438=-1	;ROUGH-OUT TOOL ~
Q351=+1	;CLIMB OR UP-CUT



## 10.7 Cycle 277 OCM CHAMFERING (option 167)

### ISO programming

#### G277

### 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:

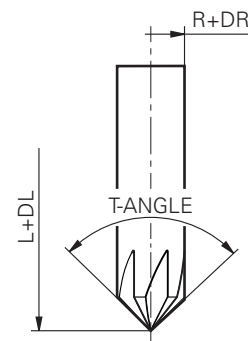
- **L + DL**: Overall length up to the theoretical tip
- **R + DR**: Definition of the overall tool radius
- **T-ANGLE**: Point angle of the tool

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 sequence

- 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**



## Notes

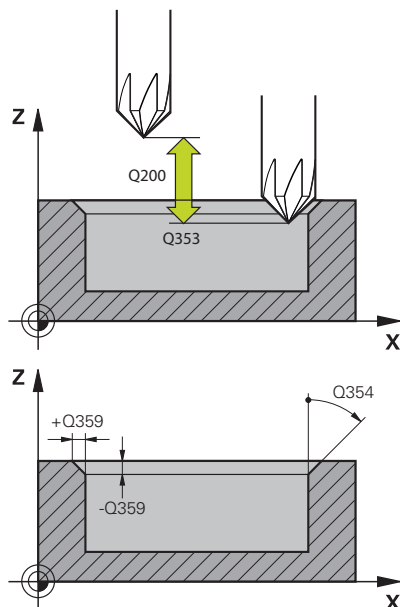
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for chamfering. The starting point depends on the available space.
- Measure the length of the tool up to the theoretical tool tip.
- The control monitors the tool radius. Adjacent walls machined with Cycle **271 OCM CONTOUR DATA** or with the **12xx** figure cycles will remain intact.
- Remember that the control does not monitor the theoretical tool tip for collision. In **Test Run** operating mode, the control always uses the theoretical tool tip for the simulation. This can result, for example for tools without an actual tool tip, in simulated contour damages while the NC program is actually error-free.
- Keep in mind that the effective tool radius must be smaller or equal to the radius of the rough-out tool. Otherwise, the control might not be able to completely chamfer all edges. The effective tool radius is the radius of the cutting length of the tool. The effective tool radius is calculated from **Q353 DEPTH OF TOOL TIP** and **T-ANGLE**.

### Note on programming

- 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.

## Cycle parameters

### Help graphic



### Parameter

#### Q353 Depth of tool tip?

Distance between theoretical tool tip and workpiece surface coordinate. This value has an incremental effect.

Input: **-999.9999...-0.0001**

#### Q359 Width of chamfer (-/+)?

Width or depth of chamfer:

-: Depth of chamfer

+: Width of chamfer

This value has an incremental effect.

Input: **-999.9999...+999.9999**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

#### Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min for positioning

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

#### Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

#### Q438 or QS438 Number/name of rough-out tool?

Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field.

**-1**: The control assumes that the tool last used is the rough-out tool (default behavior).

Input: **-1...+32767.9** or max. **255** characters

#### 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 of a **GLOBAL DEF** block (If you enter 0, climb milling is performed)

Input: **-1, 0, +1** or **PREDEF**

## Help graphic

## Parameter

**Q354 Angle of chamfer?**

Angle of the chamfer

**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: **0...89**

## Example

11 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 figures

### Fundamentals

The control provides cycles for standard figures. You can program these standard figures as pockets, islands, or boundaries.

**The cycles offer the following advantages:**

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

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

**The control offers the following cycles for standard figures:**

- **1271 OCM RECTANGLE**, see Page 343
- **1272 OCM CIRCLE**, see Page 346
- **1273 OCM SLOT / RIDGE**, see Page 349
- **1278 OCM POLYGON**, see Page 352

**The control provides the following cycles for figure boundaries:**

- **1281 OCM RECTANGLE BOUNDARY**, see Page 355
- **1282 OCM CIRCLE BOUNDARY**, see Page 357

## Tolerances

The control allows you to store tolerances in the following cycles and cycle parameters:

Cycle number	Parameter
1271 OCM RECTANGLE	Q218 FIRST SIDE LENGTH, Q219 2ND SIDE LENGTH
1272 OCM CIRCLE	Q223 CIRCLE DIAMETER
1273 OCM SLOT / RIDGE	Q219 SLOT WIDTH, Q218 SLOT LENGTH
1278 OCM POLYGON	Q571 REF-CIRCLE DIAMETER

You can define the following tolerances:

Tolerances	Example	Manufacturing dimension
Dimensions	10+0.01-0.015	9.9975
DIN EN ISO 286-2	10H7	10.0075
DIN ISO 2768-1	10m	10.0000



Pay attention to capitalization when entering tolerances.

Proceed as follows:

- ▶ Start the cycle definition
- ▶ Define the cycle parameters
- ▶ Press the **ENTER TEXT** soft key
- ▶ Enter a nominal dimension including tolerance



If you program an incorrect tolerance, the control interrupts machining with an error message.

## 10.9 Cycle 1271 OCM RECTANGLE (option 167)

### ISO programming

#### G1271

### Application

Use the figure cycle **1271 OCM RECTANGLE** to program a rectangle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program tolerances for the lengths.

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

- Cycle **1271 OCM RECTANGLE**
  - If you program **Q650=1** (figure 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

### Notes

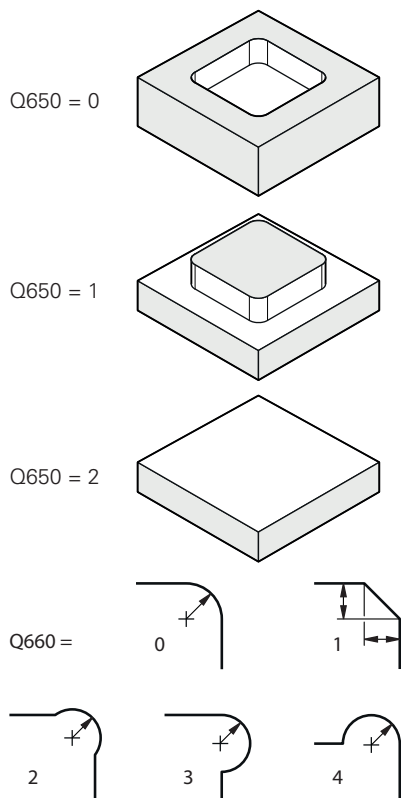
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1271** are valid for the OCM machining cycles **272** to **274** and **277**.

### Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

## Cycle parameters

### Help graphic



### Parameter

#### Q650 Type of figure?

Geometry of the figure:

**0:** Pocket

**1:** Island

**2:** Boundary for face milling

Input: **0, 1, 2**

#### Q218 First side length?

Length of the first side of the figure, parallel to the main axis. This value has an incremental effect. You can program a tolerance if needed.

**Further information:** "Tolerances", Page 342

Input: **0...99999.9999**

#### Q219 Second side length?

Length of the 2nd side of the figure, parallel to the secondary axis. This value has an incremental effect. You can program a tolerance if needed.

**Further information:** "Tolerances", Page 342

Input: **0...99999.9999**

#### Q660 Type of corners?

Geometry of the corners:

**0:** Radius

**1:** Chamfer

**2:** Milling corners in the main and secondary axis directions

**3:** Milling corners in the main axis direction

**4:** Milling corners in the secondary axis direction

Input: **0, 1, 2, 3, 4**

#### Q220 Corner radius?

Radius or chamfer of the corner of the figure

Input: **0...99999.9999**

#### Q367 Position of pocket (0/1/2/3/4)?

Position of the figure relative to the position of the tool when the cycle is called:

**0:** Tool position = Center of figure

**1:** Tool position = Lower left corner

**2:** Tool position = Lower right corner

**3:** Tool position = Upper right corner

**4:** Tool position = Upper left corner

Input: **0, 1, 2, 3, 4**

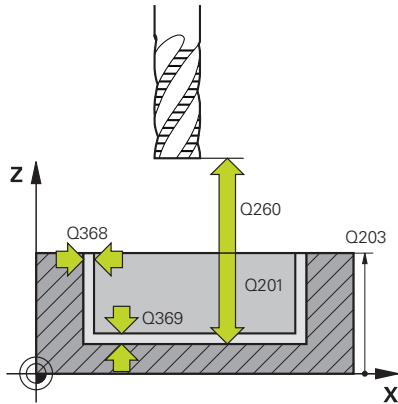
#### Q224 Angle of rotation?

Angle by which the figure is rotated. The center of rotation is at the center of the figure. The value has an absolute effect.

Input: **-360.000...+360.000**



## Help graphic



## Parameter

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q201 Depth?**

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

**Q368 Finishing allowance for side?**

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q260 Clearance height?**

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

**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: **0.05...0.99**

## Example

<b>11 CYCL DEF 1271 OCM RECTANGLE ~</b>	
<b>Q650=+1</b>	<b>;FIGURE TYPE ~</b>
<b>Q218=+60</b>	<b>;FIRST SIDE LENGTH ~</b>
<b>Q219=+40</b>	<b>;2ND SIDE LENGTH ~</b>
<b>Q660=+0</b>	<b>;CORNER TYPE ~</b>
<b>Q220=+0</b>	<b>;CORNER RADIUS ~</b>
<b>Q367=+0</b>	<b>;POCKET POSITION ~</b>
<b>Q224=+0</b>	<b>;ANGLE OF ROTATION ~</b>
<b>Q203=+0</b>	<b>;SURFACE COORDINATE ~</b>
<b>Q201=-10</b>	<b>;DEPTH ~</b>
<b>Q368=+0</b>	<b>;ALLOWANCE FOR SIDE ~</b>
<b>Q369=+0</b>	<b>;ALLOWANCE FOR FLOOR ~</b>
<b>Q260=+50</b>	<b>;CLEARANCE HEIGHT ~</b>
<b>Q578=+0.2</b>	<b>;INSIDE CORNER FACTOR</b>

## 10.10 Cycle 1272 OCM CIRCLE (option 167)

### ISO programming

#### G1272

### Application

Use figure cycle **1272 OCM CIRCLE** to program a circle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the diameter.

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

### Notes

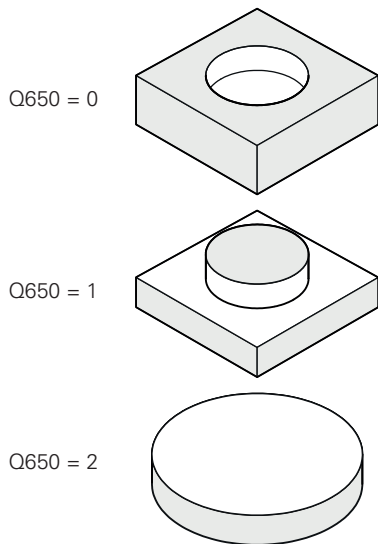
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1272** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1272** are valid for the OCM machining cycles **272** to **274** and **277**.

### Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

## Cycle parameters

### Help graphic



### Parameter

#### Q650 Type of figure?

Geometry of the figure:

- 0:** Pocket
- 1:** Island
- 2:** Boundary for face milling

Input: **0, 1, 2**

#### Q223 Circle diameter?

Diameter of the finished circle. You can program a tolerance if needed.

**Further information:** "Tolerances", Page 342

Input: **0...99999.9999**

#### Q367 Position of pocket (0/1/2/3/4)?

Position of the figure relative to the position of the tool during the cycle call:

- 0:** Tool pos. = Center of figure
- 1:** Tool pos. = Quadrant transition at 90°
- 2:** Tool pos. = Quadrant transition at 0°
- 3:** Tool pos. = Quadrant transition at 270°
- 4:** Tool pos. = Quadrant transition at 180°

Input: **0, 1, 2, 3, 4**

#### Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q201 Depth?

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

#### Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q369 Finishing allowance for floor?

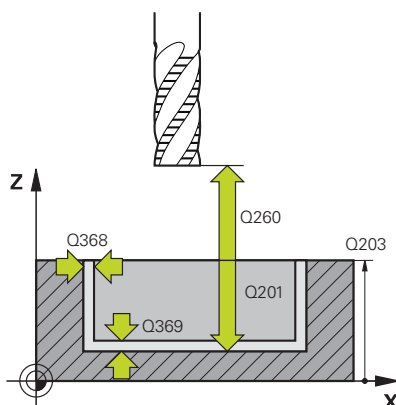
Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

#### Q260 Clearance height?

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic	Parameter
	<p><b>Q578 Radius factor on inside corners?</b></p> <p>The minimum radius of a circular pocket results from the tool radius plus the product of tool radius and <b>Q578</b>.</p> <p>Input: <b>0.05...0.99</b></p>

Example

11 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 Cycle 1273 OCM SLOT / RIDGE (option 167)

### ISO programming G1273

#### Application

Use figure cycle **1273 OCM SLOT / RIDGE** to program a slot or a ridge. This figure cycle also allows you to program a boundary for face milling. In addition, you can program a tolerance for the width and the length.

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

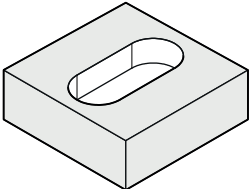
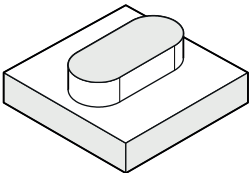
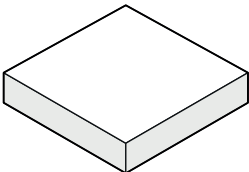
#### Notes

- 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**.

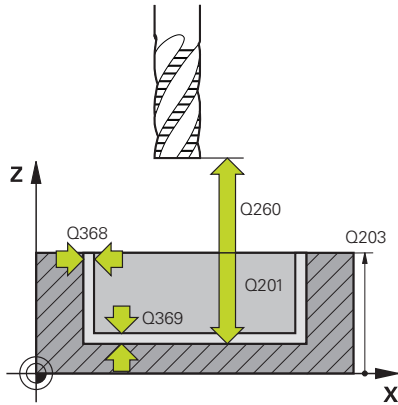
#### Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters

Help graphic	Parameter
<div><div>Q650 = 0</div></div>	<p><b>Q650 Type of figure?</b> Geometry of the figure: <b>0:</b> Pocket <b>1:</b> Island <b>2:</b> Boundary for face milling Input: <b>0, 1, 2</b></p>
<div><div>Q650 = 1</div></div>	<p><b>Q219 Width of slot?</b> Width of the slot or ridge, parallel to the secondary axis of the working plane. This value has an incremental effect. You can program a tolerance if needed. <b>Further information:</b> "Tolerances", Page 342 Input: <b>0...99999.9999</b></p>
<div><div>Q650 = 2</div></div>	<p><b>Q218 Length of slot?</b> Length of the slot or ridge, parallel to the main axis of the working plane. This value has an incremental effect. You can program a tolerance if needed. <b>Further information:</b> "Tolerances", Page 342 Input: <b>0...99999.9999</b></p>
	<p><b>Q367 Position of slot (0/1/2/3/4)?</b> Position of the figure relative to the position of the tool when the cycle is called: <b>0:</b> Tool position = Center of figure <b>1:</b> Tool position = Left end of figure <b>2:</b> Tool position = Center of left figure arc <b>3:</b> Tool position = Center of right figure arc <b>4:</b> Tool position = Right end of figure Input: <b>0, 1, 2, 3, 4</b></p>
	<p><b>Q224 Angle of rotation?</b> Angle by which the figure is rotated. The center of rotation is at the center of the figure. The value has an absolute effect. Input: <b>-360.000...+360.000</b></p>

## Help graphic



## Parameter

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q201 Depth?**

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

**Q368 Finishing allowance for side?**

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q260 Clearance height?**

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

**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: **0.05...0.99**

## Example

11 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 Cycle 1278 OCM POLYGON (option 167)

### ISO programming

#### G1278

### Application

Use figure cycle **1278 OCM POLYGON** to program a polygon. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the reference diameter.

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

### Notes

- 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**.

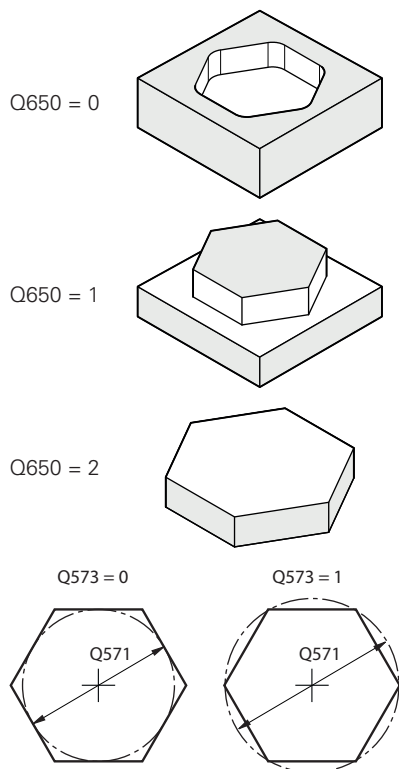
### Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.



## Cycle parameters

### Help graphic



### Parameter

#### Q650 Type of figure?

Geometry of the figure:

**0:** Pocket

**1:** Island

**2:** Boundary for face milling

Input: **0, 1, 2**

#### Q573 Inscr.circle/circumcircle (0/1)?

Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle:

**0:** Dimension is referenced to the inscribed circle

**1:** Dimension is referenced to the circumcircle

Input: **0, 1**

#### Q571 Reference circle diameter?

Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed.

**Further information:** "Tolerances", Page 342

Input: **0...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: **3...30**

#### Q660 Type of corners?

Geometry of the corners:

**0:** Radius

**1:** Chamfer

Input: **0, 1**

#### Q220 Corner radius?

Radius or chamfer of the corner of the figure

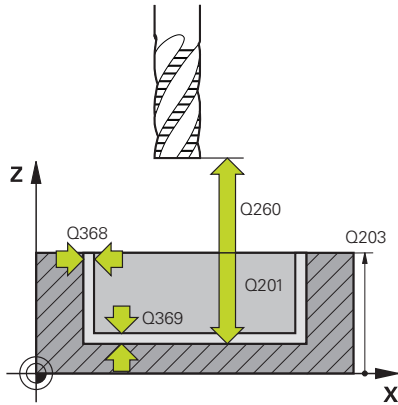
Input: **0...99999.9999**

#### Q224 Angle of rotation?

Angle by which the figure is rotated. The center of rotation is at the center of the figure. The value has an absolute effect.

Input: **-360.000...+360.000**

## Help graphic



## Parameter

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q201 Depth?**

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

**Q368 Finishing allowance for side?**

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

**Q369 Finishing allowance for floor?**

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

**Q260 Clearance height?**

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

**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: **0.05...0.99**

## Example

11 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 Cycle 1281 OCM RECTANGLE BOUNDARY (option 167)

ISO programming  
G1281

### Application

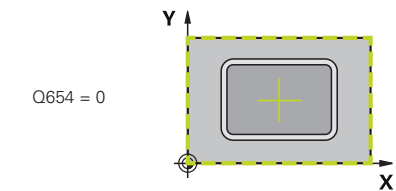
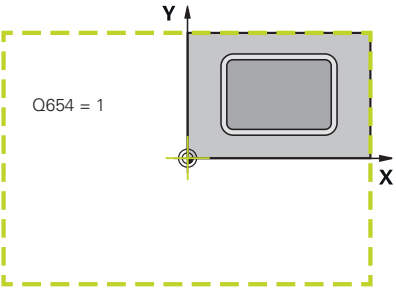
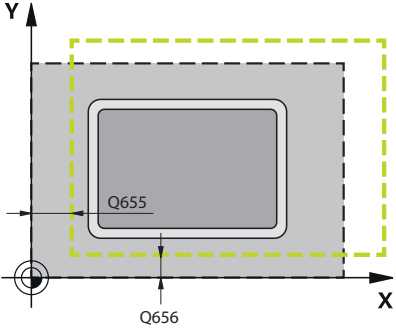
Use Cycle **1281 OCM RECTANGLE BOUNDARY** to program a rectangular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

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

### Notes

- 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

Help graphic	Parameter
	<p><b>Q651 Length of major axis?</b> Length of the first side of the boundary, parallel to the main axis Input: <b>0.001...9999.999</b></p>
	<p><b>Q652 Length of minor axis?</b> Length of the second side of the boundary, parallel to the secondary axis Input: <b>0.001...9999.999</b></p>
	<p><b>Q654 Position reference for figure?</b> Specify the position reference for the center: <b>0:</b> The center of the boundary is referenced to the center of the contour <b>1:</b> The center of the boundary is referenced to the datum Input: <b>0, 1</b></p>
	<p><b>Q655 Shift in major axis?</b> Shift of the rectangle boundary along the main axis Input: <b>-999.999...999.999</b></p>
	<p><b>Q656 Shift in minor axis?</b> Shift of the rectangle boundary along the secondary axis Input: <b>-999.999...999.999</b></p>

### Example

11 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 Cycle 1282 OCM CIRCLE BOUNDARY (option 167)

ISO programming  
G1282

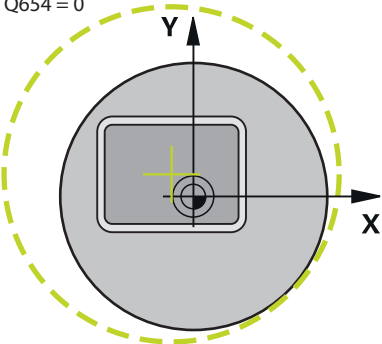
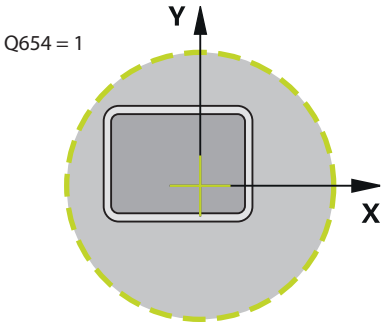
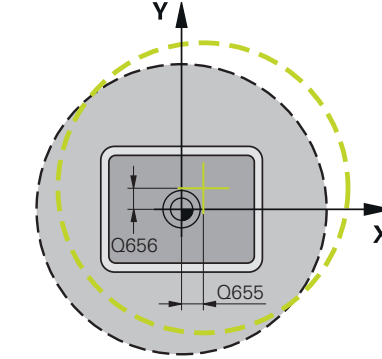
### Application

Cycle **1282 OCM CIRCLE BOUNDARY** allows you 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 figure. The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE = 0** (pocket) or **= 1** (island) in an OCM standard shape cycle.

### Notes

- 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

Help graphic	Parameter
<p>Q654 = 0</p> 	<p><b>Q653 Diameter?</b> Diameter of the circular bounding frame Input: <b>0.001...9999.999</b></p>
<p>Q654 = 1</p> 	<p><b>Q654 Position reference for figure?</b> Specify the position reference for the center: <b>0:</b> The center of the boundary is referenced to the center of the contour <b>1:</b> The center of the boundary is referenced to the datum Input: <b>0, 1</b></p>
	<p><b>Q655 Shift in major axis?</b> Shift of the rectangle boundary along the main axis Input: <b>-999.999...999.999</b></p> <p><b>Q656 Shift in minor axis?</b> Shift of the rectangle boundary along the secondary axis Input: <b>-999.999...999.999</b></p>

Example

11 CYCL DEF 1282 OCM CIRCLE BOUNDARY ~	
Q653=+50	;DIAMETER ~
Q654=+0	;POSITION REFERENCE ~
Q655=+0	;SHIFT 1 ~
Q656=+0	;SHIFT 2

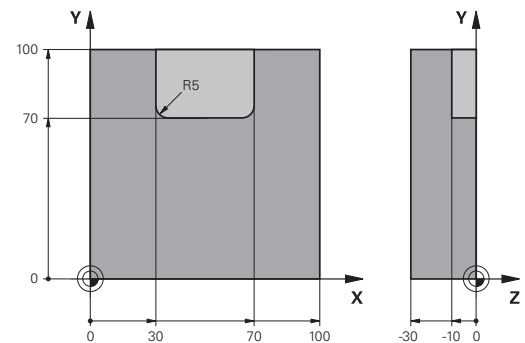
## 10.15 Programming examples

### 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	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S8000 F1500	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF ~	
P1 = LBL 1 I2 = LBL;2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
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 ~	
Q569=+1 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
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	
8 CYCL CALL		; Cycle call
9 TOOL CALL 4 Z S8000 F1500		; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3		
11 CYCL DEF 272 OCM ROUGHING ~		
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 ~	
Q438=+10	;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	
12 CYCL CALL		; Cycle call
13 TOOL CALL 23 Z S10000 F2000		; Tool call (diameter: 6 mm)
14 L Z+100 R0 FMAX M3		
15 CYCL DEF 273 OCM FINISHING FLOOR ~		
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 ~	
Q595=+1	;STRATEGY ~	
Q577=+0.2	;APPROACH RADIUS FACTOR	
16 CYCL CALL		; Cycle call
17 CYCL DEF 274 OCM FINISHING SIDE ~		
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 ~	
Q438=-1	;ROUGH-OUT TOOL ~	
Q351=+1	;CLIMB OR UP-CUT	
18 CYCL CALL		; Cycle call
19 M30		; End of program
20 LBL 1		; Contour subprogram 1
21 L X+0 Y+0		
22 L X+100		



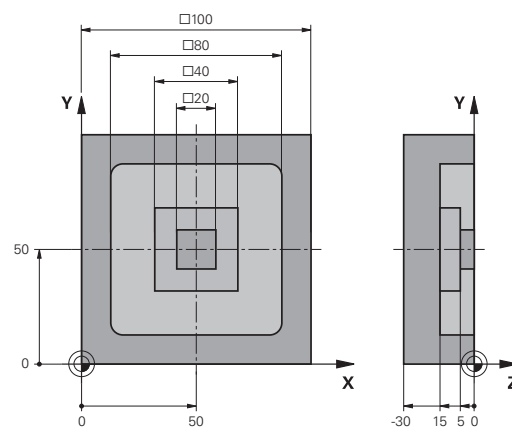
23 L Y+100	
24 L X+0	
25 L Y+0	
26 LBL 0	
27 LBL 2	; Contour subprogram 2
28 L X+0 Y+0	
29 L X+100	
30 L Y+100	
31 L X+70	
32 L Y+70	
33 RND R5	
34 L X+30	
35 RND R5	
36 L Y+100	
37 L X+0	
38 L Y+0	
39 LBL 0	
40 END PGM OCM_POCKET MM	

### 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	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 5 Z S8000 F1500	; Tool call (diameter: 10 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF ~	
P1 = LBL 1 I2 = LBL;2 I3 = LBL 3 DEPTH5	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-15 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+0 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
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	
8 CYCL CALL	; Cycle call
9 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
10 L Z+100 R0 FMAX M3	

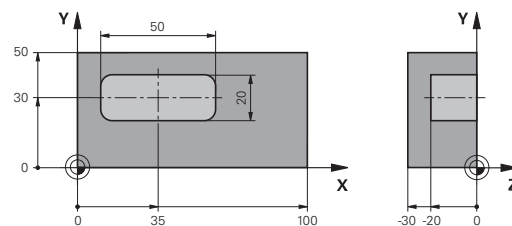
11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385= AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 CYCL CALL	; Cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
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 ~	
Q438=+5 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
14 CYCL CALL	; Cycle call
15 M30	; End of program
16 LBL 1	; Contour subprogram 1
17 L X-40 Y-40	
18 L X+40	
19 L Y+40	
20 L X-40	
21 L Y-40	
22 LBL 0	
23 LBL 2	; Contour subprogram 2
24 L X-10 Y-10	
25 L X+10	
26 L Y+10	
27 L X-10	
28 L Y-10	
29 LBL 0	
30 LBL 3	; Contour subprogram 3
31 L X-20 Y-20	
32 L Y+20	
33 L X+20	
34 L Y-20	
35 L X-20	
36 LBL 0	
37 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	
2 BLK FORM 0.2 X+100 Y+50 Z+2	
3 TOOL CALL 6 Z S5000 F3000	; Tool call (diameter: 12 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF ~	
P1 = LBL 1 I2 = LBL;1 DEPTH2 ~	
P3 = LBL 2 ;	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
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	
7 CYCL DEF 272 OCM ROUGHING ~	
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=+1 ;INFEEED STRATEGY	
8 L X+0 Y+0 R0 FMAX M99	; Cycle call
9 TOOL CALL 4 Z S6000 F4000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	

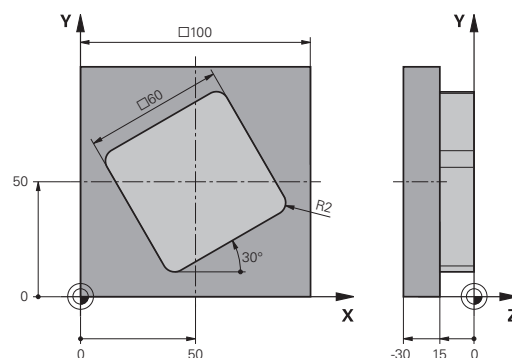
11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+25 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207= 6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+6 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEEED STRATEGY	
12 L X+0 Y+0 R0 FMAX M99	; Cycle call
13 M30	; End of program
14 LBL 1	; Contour subprogram 1
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 2	; Contour subprogram 2
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 figure 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	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 1271 OCM RECTANGLE ~	
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 ~	
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 ~	
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	;INFEED STRATEGY	
8 L X+50 Y+50 R0 FMAX M99		; Positioning and cycle call
9 TOOL CALL 24 Z S10000 F2000		; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3		
11 CYCL DEF 273 OCM FINISHING FLOOR ~		
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=+4	;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 ~		
Q338=+15	;INFEED FOR FINISHING ~	
Q385= AUTO	;FINISHING FEED RATE ~	
Q253= AUTO	;F PRE-POSITIONING ~	
Q200=+2	;SET-UP CLEARANCE ~	
Q14=+0	;ALLOWANCE FOR SIDE ~	
Q438=+4	;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
	Cycle 27 CYLINDER SURFACE (option 8) <ul style="list-style-type: none"> <li>■ Milling of guide slots on the cylinder surface</li> <li>■ Slot width is equal to tool radius</li> </ul>	371
	Cycle 28 CYLINDRICAL SURFACE SLOT (option 8) <ul style="list-style-type: none"> <li>■ Milling of guide slots on the cylinder surface</li> <li>■ Input of the slot width</li> </ul>	374
	Cycle 29 CYL SURFACE RIDGE (option 8) <ul style="list-style-type: none"> <li>■ Milling of a ridge on the cylinder surface</li> <li>■ Input of the ridge width</li> </ul>	379
	Cycle 39 CYL. SURFACE CONTOUR (option 8) <ul style="list-style-type: none"> <li>■ Milling of a contour on the cylinder surface</li> </ul>	383

## 11.2 Cycle 27 CYLINDER SURFACE (option 8)

### ISO programming

G127

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to program a contour in two dimensions and then transfer it onto a cylindrical surface. 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.

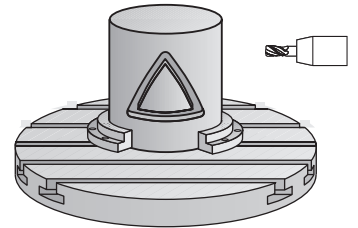
The dimensions in the rotary axis (X coordinates) can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition using **Q17**.

### Cycle sequence

- 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.



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



## Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- 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.



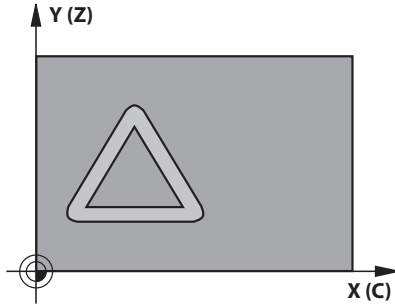
The machining time can increase if the contour consists of many non-tangential contour elements.

## Notes on programming

- 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 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

### Help graphic



### Parameter

#### Q1 Milling depth?

Distance between cylindrical surface and contour floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q3 Finishing allowance for side?

Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q6 Set-up clearance?

Distance between the tool face and the cylindrical surface. This value has an incremental effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

#### Q10 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q11 Feed rate for plunging?

Traversing feed rate in the spindle axis

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q12 Feed rate for roughing?

Traversing feed rate in the working plane

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q16 Cylinder radius?

Radius of the cylinder on which the contour will be machined.

Input: **0...99999.9999**

#### Q17 Dimension type? deg=0 MM/INCH=1

Program the rotary axis coordinates in degrees or mm (inches) in the subprogram.

Input: **0, 1**

### Example

11 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-20	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+0	;SET-UP CLEARANCE ~
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q16=+0	;RADIUS ~
Q17=+0	;TYPE OF DIMENSION

## 11.3 Cycle 28 CYLINDRICAL SURFACE SLOT (option 8)

ISO programming  
G128

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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.

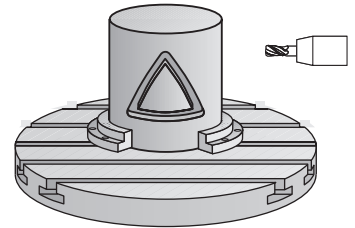
Program the center path of the contour together with the tool radius compensation. With the radius compensation you specify whether the control cuts the slot with climb milling or up-cut milling.

### Cycle sequence

- 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.



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



## Notes



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** machine 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, check 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.
- 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 machining time can increase if the contour consists of many non-tangential contour elements.

## Notes on programming

- 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 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.

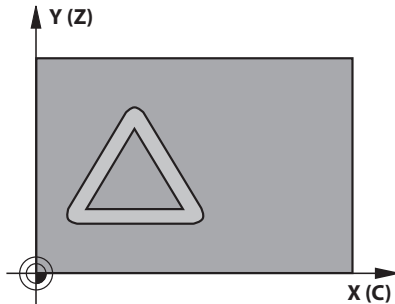
**Note regarding machine parameters**

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line



## Cycle parameters

### Help graphic



### Parameter

#### Q1 Milling depth?

Distance between cylindrical surface and contour floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q3 Finishing allowance for side?

Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q6 Set-up clearance?

Distance between the tool face and the cylindrical surface. This value has an incremental effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

#### Q10 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

#### Q11 Feed rate for plunging?

Traversing feed rate in the spindle axis

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q12 Feed rate for roughing?

Traversing feed rate in the working plane

Input: **0...99999.9999** or **FAUTO, FU, FZ**

#### Q16 Cylinder radius?

Radius of the cylinder on which the contour will be machined.

Input: **0...99999.9999**

#### Q17 Dimension type? deg=0 MM/INCH=1

Program the rotary axis coordinates in degrees or mm (inches) in the subprogram.

Input: **0, 1**

#### Q20 Slot width?

Width of the slot to be machined

Input: **-99999.9999...+99999.9999**

## Help graphic

## Parameter

**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: **0...9.9999**

## Example

11 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~	
Q1=-20	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+2	;SET-UP CLEARANCE ~
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q16=+0	;RADIUS ~
Q17=+0	;TYPE OF DIMENSION ~
Q20=+0	;SLOT WIDTH ~
Q21=+0	;TOLERANCE

## 11.4 Cycle 29 CYL SURFACE RIDGE (option 8)

### ISO programming

G129

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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.

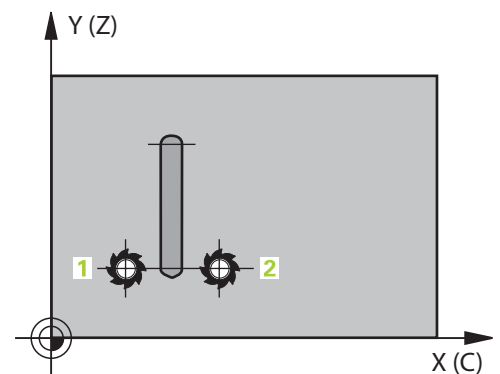
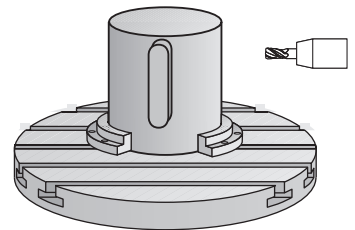
At the ends of the ridge, the control will always add a semi-circle whose radius corresponds to half the ridge width.

### Cycle sequence

- 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.



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



## Notes



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** machine 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.
- 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.

#### Notes on programming

- 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 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

Help graphic	Parameter
	<b>Q1 Milling depth?</b> Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q3 Finishing allowance for side?</b> Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q6 Set-up clearance?</b> Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b> or <b>PREDEF</b>
	<b>Q10 Plunging depth?</b> Tool infeed per cut. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q11 Feed rate for plunging?</b> Traversing feed rate in the spindle axis Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q12 Feed rate for roughing?</b> Traversing feed rate in the working plane Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q16 Cylinder radius?</b> Radius of the cylinder on which the contour will be machined. Input: <b>0...99999.9999</b>
	<b>Q17 Dimension type? deg=0 MM/INCH=1</b> Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: <b>0, 1</b>
	<b>Q20 Ridge width?</b> Width of the ridge to be machined Input: <b>-99999.9999...+99999.9999</b>

**Example**

11 CYCL DEF 29 CYL SURFACE RIDGE ~	
Q1=-20	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+2	;SET-UP CLEARANCE ~
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q16=+0	;RADIUS ~
Q17=+0	;TYPE OF DIMENSION ~
Q20=+0	;RIDGE WIDTH

## 11.5 Cycle 39 CYL. SURFACE CONTOUR (option 8)

ISO programming  
G139

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle, the 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.

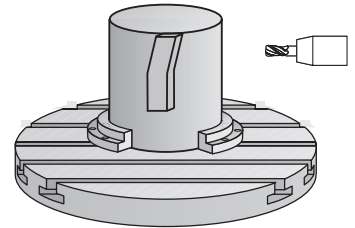
Unlike in Cycles **28** and **29**, in the contour subprogram, you define the contour actually to be machined.

### Cycle sequence

- 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 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**. A finishing allowance programmed for the side is taken into account. The approach behavior depends on the machine parameter **apprDepCylWall** (no. 201004)
- 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.



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



## Notes



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** machine 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.
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.



- 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.

## Notes on programming

- 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 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.

## Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line



## Cycle parameters

Help graphic	Parameter
	<b>Q1 Milling depth?</b> Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q3 Finishing allowance for side?</b> Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q6 Set-up clearance?</b> Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b> or <b>PREDEF</b>
	<b>Q10 Plunging depth?</b> Tool infeed per cut. This value has an incremental effect. Input: <b>-99999.9999...+99999.9999</b>
	<b>Q11 Feed rate for plunging?</b> Traversing feed rate in the spindle axis Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q12 Feed rate for roughing?</b> Traversing feed rate in the working plane Input: <b>0...99999.9999</b> or <b>FAUTO, FU, FZ</b>
	<b>Q16 Cylinder radius?</b> Radius of the cylinder on which the contour will be machined. Input: <b>0...99999.9999</b>
	<b>Q17 Dimension type? deg=0 MM/INCH=1</b> Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: <b>0, 1</b>

### Example

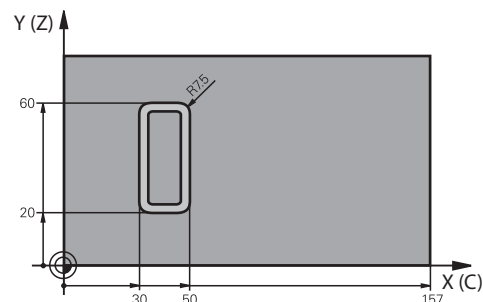
11 CYCL DEF 39 CYL. SURFACE CONTOUR ~	
Q1=-20	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+2	;SET-UP CLEARANCE ~
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q16=+0	;RADIUS ~
Q17=+0	;TYPE OF DIMENSION

## 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



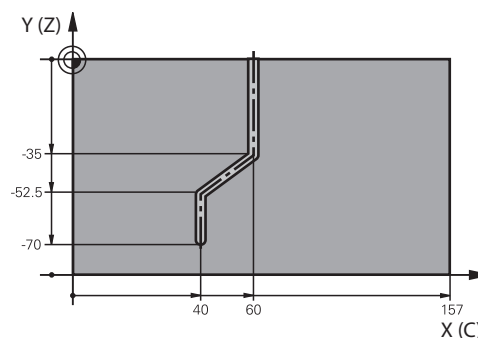
0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call (diameter: 7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-7	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+2	;SET-UP CLEARANCE ~
Q10=-4	;PLUNGING DEPTH ~
Q11=+100	;FEED RATE FOR PLNGNG ~
Q12=+250	;FEED RATE F. ROUGHNG ~
Q16=+25	;RADIUS ~
Q17=+1	;TYPE OF DIMENSION
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram
13 L X+40 Y-20 RL	; Rotary axis data in mm (Q17 = 1)
14 L X+50	
15 RND R7.5	
16 L Y-60	
17 RND 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 5 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 path of the tool center in the contour subprogram



0 BEGIN PGM 4 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call, tool axis (Z), diameter (7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~	
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	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram, description of the path of the tool center
13 L X+60 Y+0 RL	; Rotary axis data in mm (Q17 = 1)
14 L Y-35	
15 L X+40 Y-52.5	
16 L X-70	
17 LBL 0	
18 END PGM 4 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.

#### Program structure: Machining with SL Cycles and complex contour formula

```

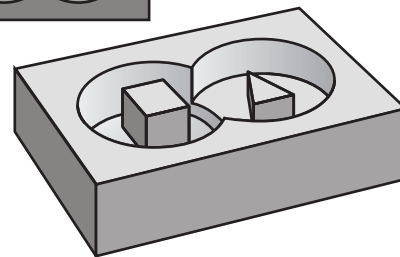
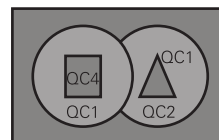
0 BEGIN CONT MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA
...
8 CYCL DEF 21 ROUGH-OUT
...
9 CYCL CALL
...
13 CYCL DEF 23 FLOOR FINISHING
...
14 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING
...
17 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 END PGM CONT MM

```



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.



**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 MODEL MM
```

```
1 DECLARE CONTOUR QC1 = "120"
```

```
2 DECLARE CONTOUR QC2 = "121" DEPTH15
```

```
3 DECLARE CONTOUR QC3 = "122" DEPTH10
```

```
4 DECLARE CONTOUR QC4 = "123" DEPTH5
```

```
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
```

```
6 END PGM MODEL MM
```

```
0 BEGIN PGM 120 MM
```

```
1 CC X+75 Y+50
```

```
2 LP PR+45 PA+0
```

```
3 CP IPA+360 DR+
```

```
4 END PGM 120 MM
```

```
0 BEGIN PGM 121 MM
```

```
...
```

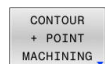
## Selecting an NC program with contour definitions

With the **SEL CONTOUR** function, you select an NC program with contour definitions, from which the control extracts the contour descriptions:

Proceed as follows:



- ▶ Press the **SPEC FCT** key

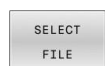


- ▶ Press the **CONTOUR AND POINT MACHINING** soft key



- ▶ 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.

Proceed as follows:

- SPEC  
FCT

▶ Press the **SPEC FCT** key
- CONTOUR  
+ POINT  
MACHINING

▶ Press the **CONTOUR AND POINT MACHINING** soft key
- DECLARE  
CONTOUR

▶ Press the **DECLARE CONTOUR** soft key.
- SELECT  
FILE

▶ Press the **SELECT FILE** soft key and select the desired NC program
- SELECT  
FILE

▶ Define a separate depth for the selected contour
- SELECT  
FILE

▶ Press the **END** key

or



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 400







## 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

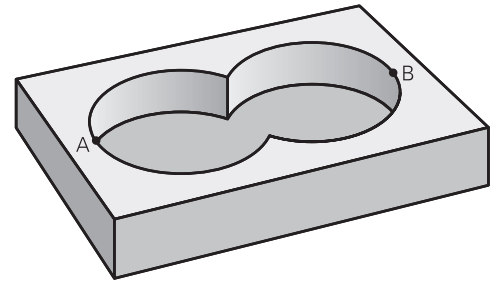
The control displays the following soft keys:

Soft key	Mathematical function
	<b>Intersected with</b> e.g. $QC10 = QC1 \& QC5$
	<b>Joined with</b> e.g. $QC25 = QC7   QC18$
	<b>joined with, but without intersection</b> e.g. $QC12 = QC5 \wedge QC25$
	<b>without</b> e.g. $QC25 = QC1 \setminus QC2$
	<b>Open parenthesis</b> e.g. $QC12 = QC1 \& (QC2   QC3)$
	<b>Close parenthesis</b> e.g. $QC12 = QC1 \& (QC2   QC3)$
	<b>Define single contour</b> e.g. $QC12 = QC1$

## 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 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 MM
```

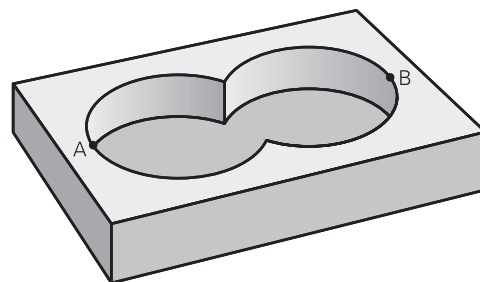
#### Contour description program 2: pocket B

```
0 BEGIN PGM POCKET2 MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET2 MM
```

**Area of inclusion**

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

- Areas 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:**

```
* - ...
```

```
21 DECLARE CONTOUR QC1 = "POCKET.H"
```

```
22 DECLARE CONTOUR QC2 = "POCKET2.H"
```

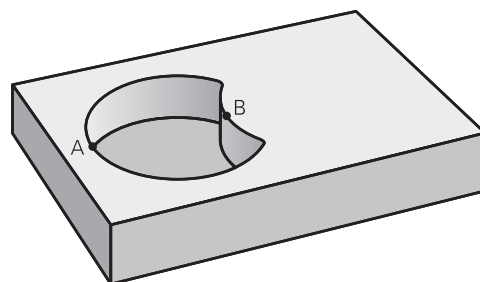
```
23 QC10 = QC1 | QC2
```

```
* - ...
```

**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:**

```
* - ...
```

```
21 DECLARE CONTOUR QC1 = "POCKET.H"
```

```
22 DECLARE CONTOUR QC2 = "POCKET2.H"
```

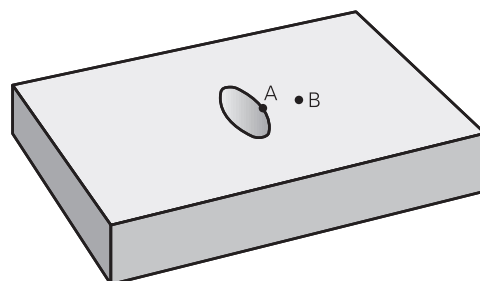
```
23 QC10 = QC1 \ QC2
```

```
* - ...
```

**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:**

```
* - ...
```

```
21 DECLARE CONTOUR QC1 = "POCKET.H"
```

```
22 DECLARE CONTOUR QC2 = "POCKET2.H"
```

```
23 QC10 = QC1 & QC2
```

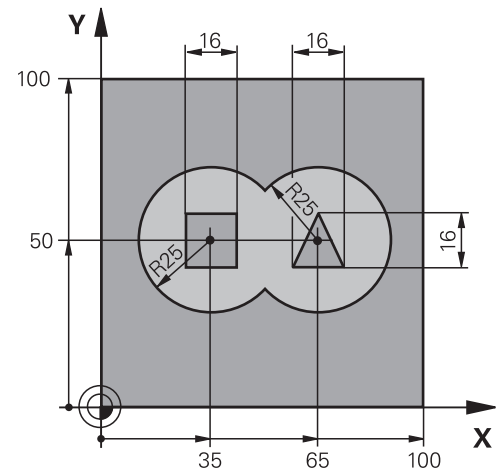
```
* - ...
```

## Machining contours with SL or OCM cycles



The entire contour is machined with the SL cycles (see "Overview", Page 262) or the OCM cycles (see "Overview", Page 311).

### 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 5 Z S2500	; Tool call: roughing cutter
4 L Z+250 R0 FMAX M3	; Retract the tool
5 SEL CONTOUR "MODEL"	; Specify the contour definition program
6 CYCL DEF 20 CONTOUR DATA ~	; Define the 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		; Cycle call: rough-out
9 TOOL CALL 23 Z S5000		; Tool call: finishing cutter
10 L Z+250 R0 FMAX M3		
11 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	
12 CYCL CALL		; Cycle call: floor finishing
13 CYCL DEF 24 SIDE FINISHING ~		; Cycle definition: side finishing
Q9=+1	;ROTATIONAL DIRECTION ~	
Q10=-10	;PLUNGING DEPTH ~	
Q11=+100	;FEED RATE FOR PLNGNG ~	
Q12=+400	;FEED RATE F. ROUGHNG ~	
Q14=+0	;ALLOWANCE FOR SIDE	
14 CYCL CALL		; Cycle call: side finishing
15 L Z+250 R0 FMAX		; Retract the tool, end program
16 M30		
17 END PGM CONTOUR MM		

#### Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	
1 DECLARE CONTOUR QC1 = "120"	; Define the contour label for the NC program "120"
2 Q1 = 35	; Assign the values for parameters used in PGM "121"
3 Q2 = 50	
4 Q3 = 25	
5 DECLARE CONTOUR QC2 = "121"	; Define the contour label for the NC program "121"
6 DECLARE CONTOUR QC3 = "122"	; Define the contour label for the NC program "122"
7 DECLARE CONTOUR QC4 = "123"	; Define the contour label for the NC program "123"
8 QC10 = ( QC1   QC2 ) \ QC3 \ QC4	; Contour formula
9 END PGM MODEL MM	

**Contour description program for circle at right:**

0 BEGIN PGM 120 MM	
1 CC X+65 Y+50	
2 LP PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM 120 MM	

**Contour description program for circle at left:**

0 BEGIN PGM 121 MM	
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM 121 MM	

**Contour description program for triangle at right:**

0 BEGIN PGM 122 MM	
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 122 MM	

**Contour description program for square at left:**

0 BEGIN PGM 123 MM	
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 123 MM	

## 12.2 SL or OCM cycles with simple contour formula

### Fundamentals

**Program structure: Machining with SL Cycles and simple contour formula**

```

0 BEGIN CONTDEF MM
...
5 CONTOUR DEF
...
6 CYCL DEF 20 CONTOUR DATA
...
8 CYCL DEF 21 ROUGH-OUT
...
9 CYCL CALL
...
13 CYCL DEF 23 FLOOR FINISHING
...
14 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING
...
17 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 END PGM CONTDEF MM

```

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.

### Void areas

Using optional void areas **V (void)**, you can exclude areas from machining. These area can be, for example, contours in castings or areas machined in previous steps. You can define up to five void areas.

If you are using OCM cycles, the control will plunge vertically within void areas.

If you are using SL Cycles **22** to **24**, the control will determine the plunging position, regardless of any defined void areas.

Run the simulation to verify proper behavior.



**Properties of the subcontours**

- Do not program 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 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 dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM 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 **P1**. Confirm with the **ENT** key
- ▶ Press the **POCKET (P)** soft key
- or
- ▶ Press the **ISLAND (I)** 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.
- ▶ Define void areas **V** as needed

**i** The depth of the void areas corresponds to the total depth that you define in the machining cycle.

You can enter contours in the following ways:

Soft key	Function
CONTOUR <FILE>	Define the name of the contour
SELECT FILE	or Press the <b>SELECT FILE</b> soft key
CONTOUR <FILE>=QS	Define the number of a QS parameter
CONTOUR LBL NR	Define the number of a label
CONTOUR LBL NAME	Define the name of the label
CONTOUR LBL QS	Define the number of a QS parameter for a label

### Example:

11 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 DEPTH5 V1 = LBL 3



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 262) or the OCM cycles (see "Overview", Page 311).












# 13

**Cycles: Special  
Functions**

## 13.1 Fundamentals

### Overview

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

Soft key	Cycle	Page
	Cycle 9 DWELL TIME <ul style="list-style-type: none"> <li>■ Delay execution by the programmed dwell time</li> </ul>	407
	Cycle 12 PGM CALL <ul style="list-style-type: none"> <li>■ Call any NC program</li> </ul>	408
	Cycle 13 ORIENTATION <ul style="list-style-type: none"> <li>■ Rotate spindle to a specific angle</li> </ul>	410
	Cycle 32 TOLERANCE <ul style="list-style-type: none"> <li>■ Program the permissible contour deviation for jerk-free machining operations</li> </ul>	411
	Cycle 225 ENGRAVING <ul style="list-style-type: none"> <li>■ Engrave texts on a plane surface</li> <li>■ Arranged in a straight line or along a circular arc</li> </ul>	415
	Cycle 232 FACE MILLING (option 19) <ul style="list-style-type: none"> <li>■ Face mill a level surface in multiple infeeds</li> <li>■ Selection of the milling plan</li> </ul>	422
	Cycle 238 MEASURE MACHINE STATUS (option 155) <ul style="list-style-type: none"> <li>■ Determine the current machine status or test the measuring sequence</li> </ul>	428
	Cycle 239 ASCERTAIN THE LOAD (option 143) <ul style="list-style-type: none"> <li>■ Selection for a weighing run</li> <li>■ Reset the load-dependent feedforward and controller parameters</li> </ul>	430
	Cycle 18 THREAD CUTTING <ul style="list-style-type: none"> <li>■ With controlled spindle</li> <li>■ Spindle stops at the bottom of the hole</li> </ul>	433

## 13.2 Cycle 9 DWELL TIME

### ISO programming

G4

### Application



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

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.

### Example

```
89 CYCL DEF 9.0 DWELL TIME
```

```
90 CYCL DEF 9.1 DWELL 1.5
```



### Cycle parameters

#### Help graphic

#### Parameter

##### Dwell time in secs.?

Enter the dwell time in seconds.

Input: **0...3600 s** (1 hour) in steps of 0.001 seconds

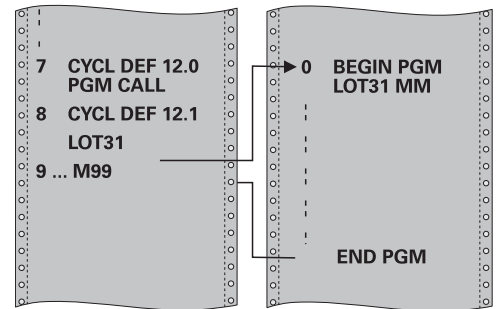
## 13.3 Cycle 12 PGM CALL

### ISO programming

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.



### Notes

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

### Notes on programming

- 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.



## Cycle parameters

Help graphic	Parameter
	<p><b>Program name</b></p> <p>Enter the name of the NC program to be called and, if necessary, the path where it is located,</p> <p>Use the Select soft key to activate the File Select dialog. Select the NC program to be called.</p> <p>The <b>SYNTAX</b> soft key allows you to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.</p> <p>If the complete path is within the quotation marks, you can use both \ and / to separate the folders and files.</p>

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

### Declare NC program 1\_Plate.h as a cycle and call it with M99

```
11 CYCL DEF 12.0 PGM CALL
```


```
12 CYCL DEF 12.1 PGM TNC:\nc_prog\demo\OCM\1_Plate.h
```

```
13 L X+20 Y+50 R0 FMAX M99
```

## 13.4 Cycle 13 ORIENTATION

ISO programming  
G36

### Application



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

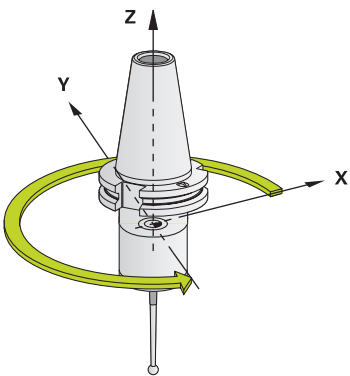
The control can control the main machine tool spindle and rotate it to a given angular position.

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.



### Notes

- 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

Help graphic	Parameter
	<b>Orientation angle</b> Enter the angle relative to the angle reference axis of the working plane. Input: <b>0...360</b>

### Example

```

11 CYCL DEF 13.0 ORIENTATION
12 CYCL DEF 13.1 ANGLE180
  
```

## 13.5 Cycle 32 TOLERANCE

### ISO programming

G62

### Application



Refer to your machine manual.  
Machine and control must be specially prepared by the machine manufacturer 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.



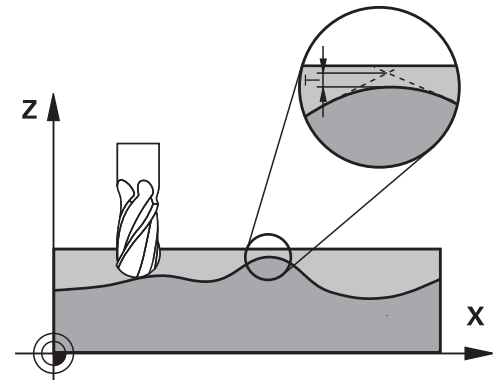
With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the control, but by the fact that, in order to machine the contour transitions very exactly, the control might have to drastically reduce the speed.

### Resetting

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

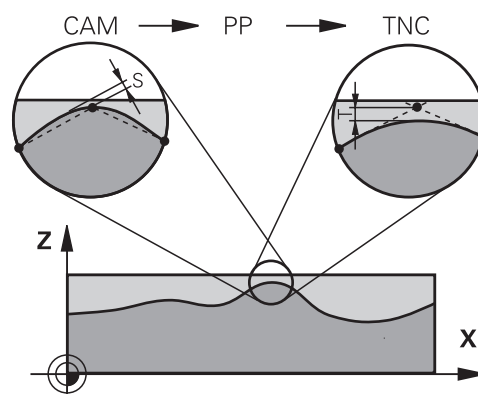
After you have reset Cycle **32**, the control reactivates the tolerance that was predefined by the machine parameters.



## 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.



## Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining modes.
- Cycle **32** is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
- In a program with millimeters set as unit of measure, the control interprets the entered tolerance value  $T$  in millimeters. In an inch program it interprets it as inches.
- If you load an NC program with Cycle **32** that contains only the **Tolerance value**  $T$  cycle parameter, the control inserts the two remaining parameters with the value 0 if required.
- As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine tool builder).
- If Cycle **32** is active, the control shows the defined cycle parameters on the **CYC** tab of the additional status display.

**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 **32G62**, 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:  
 $T \sim K \times L \times TA$   $K = 0.0175 [1/^\circ]$   
Example:  $L = 10 \text{ mm}$ ,  $TA = 0.1^\circ$ :  $T = 0.0175 \text{ mm}$

**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

$\pi$ : Circular constant (pi)

R: Major radius of the torus in mm

$T_{32}$ : Machining tolerance in mm

## Cycle parameters

Help graphic	Parameter
	<p><b>Tolerance value T</b></p> <p>Permissible contour deviation in mm (or inches with inch programming)</p> <p><b>&gt; 0:</b> If you enter a value greater than zero, the control will use the maximum permissible deviation you have specified.</p> <p><b>0:</b> If you enter zero or press the <b>NO ENT</b> key when programming, the control will use a value configured by the machine manufacturer</p> <p>Input: <b>0...10</b></p>
	<p><b>HSC mode, finishing = 0, roughing = 1</b></p> <p>Activate filter:</p> <p><b>0:</b> Milling with increased contour accuracy. The control uses internally defined finishing filter settings.</p> <p><b>1:</b> Milling with increased feed rate. The control uses internally defined roughing filter settings.</p> <p>Input: <b>0, 1</b></p>
	<p><b>Tolerance TA for rotary axes</b></p> <p>Permissible position error of rotary axes in degrees with active <b>M128 (FUNCTION TCPM)</b>. 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 exactly 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 <b>Tool Center Point (TCP)</b> 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.</p> <p><b>&gt; 0:</b> If you enter a value greater than zero, the control will use the maximum permissible deviation you have specified.</p> <p><b>0:</b> If you enter zero or press the <b>NO ENT</b> key when programming, the control will use a value configured by the machine manufacturer.</p> <p>Input: <b>0...10</b></p>

### Example

```

11 CYCL DEF 32.0 TOLERANCE
12 CYCL DEF 32.1 T0.05
13 CYCL DEF 32.2 HSC-MODE:1 TA5

```

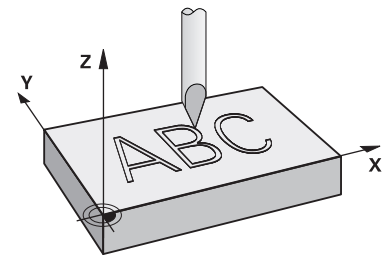
## 13.6 Cycle 225 ENGRAVING

### ISO programming

#### G225

### Application

This cycle is used to engrave texts on a flat surface of the workpiece. You can arrange the texts in a straight line or along an arc.



### Cycle sequence

- 1 If the tool is beneath **Q204 2ND SET-UP CLEARANCE**, the control will first move to the value from **Q204**.
- 2 The control positions the tool in the working plane to the starting point of the first character.
- 3 The control engraves the text.
  - If **Q202 MAX. PLUNGING DEPTH** is greater than **Q201 DEPTH**, the control will engrave each character in a single infeed motion.
  - If **Q202 MAX. PLUNGING DEPTH** is less than **Q201 DEPTH**, the control will engrave each character in several infeed motions. The control will always complete the milling of a character before machining the next one.
- 4 After the control has engraved a character, it retracts the tool to the set-up clearance **Q200** above the workpiece surface.
- 5 The process steps 2 and 3 are repeated for all characters to be engraved.
- 6 Finally, the control retracts the tool to 2nd set-up clearance **Q204**.

### Notes

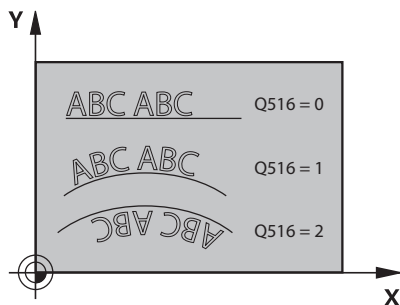
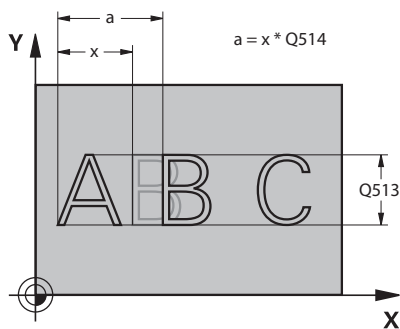
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

### Notes on programming

- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The text to be engraved can also be transferred with a string variable (**QS**).
- Parameter **Q347** influences the rotational position of the letters. If **Q374** = 0° to 180°, the characters are engraved from left to right. If **Q374** is greater than 180°, the direction of engraving is reversed.

## Cycle parameters

### Help graphic



### Parameter

#### Q500 Engraving text?

Text to be engraved within 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.

Input: Max. **255** characters

**Further information:** "Engraving system variables", Page 420

#### Q513 Character height?

Height of the characters to be engraved in mm

Input: **0...999.999**

#### 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: **0...10**

#### Q515 Font?

By default, the control uses the **DeJaVuSans** font.

#### Q516 Text on a line/on an arc(0-2)?

**0:** Engrave text in a straight line

**1:** Engrave text along an arc

**2:** Engrave text along the inside of a circular arc (circumferentially; not necessarily legible from below)

Input: **0, 1, 2**

#### Q374 Angle of rotation?

Center angle if the text is arranged on an arc. Engraving angle when text is in a straight line.

Input: **-360.000...+360.000**

#### Q517 Radius of text on an arc?

Radius of the arc in mm on which the control will engrave the text.

Input: **0...99999.9999**

#### Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

#### Q201 Depth?

Distance between workpiece surface and engraving floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**



## Help graphic

## Parameter

**Q206 Feed rate for plunging?**

Tool traversing speed in mm/min during plunging

Input: **0...99999.999** or **FAUTO, FU**

**Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q203 Workpiece surface coordinate?**

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

**Q367 Reference for text position (0-6)?**

Enter the reference for the position of the text here. Depending on whether the text will be engraved along a circular arc or in a straight line (parameter **Q516**), the following values can be entered:

**Circle****Straight line**

0 = Circle center

0 = Bottom left

1 = Bottom left

1 = Bottom left

2 = Bottom center

2 = Bottom center

3 = Bottom right

3 = Bottom right

4 = Top right

4 = Top right

5 = Top center

5 = Top center

6 = Top left

6 = Top left

7 = Center left

7 = Center left

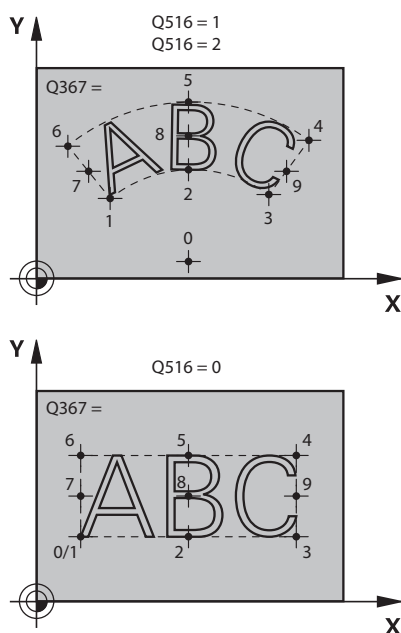
8 = Center of text

8 = Center of text

9 = Center right

9 = Center right

Input: **0...9**



## Help graphic

## Parameter

**Q574 Maximum text length?**

Enter the maximum text length. 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 > 0**, 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: **0...999.999**

**Q202 Maximum plunging depth?**

Maximum infeed depth per cut. The machining operation is performed in several steps if this value is less than **Q201**.

Input: **0...99999.9999**

## Example

11 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=+50	;CIRCLE RADIUS ~
Q207=+500	;FEED RATE MILLING ~
Q201=-2	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q367=+0	;TEXT POSITION ~
Q574=+0	;TEXT LENGTH ~
Q202=+0	;MAX. PLUNGING DEPTH

## 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:

Input	Algebraic sign
%ae	ä
%oe	ö
%ue	ü
%AE	Ä
%OE	Ö
%UE	Ü
%ss	ß
%D	ø
%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:

Input	Character
\n	Line break
\t	Horizontal tab (the tab width is permanently set to eight characters)
\v	Vertical tab (the tab width is permanently set to one line)

## 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 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**.

Input	Character
<b>%time00</b>	DD.MM.YYYY hh:mm:ss
<b>%time01</b>	D.MM.YYYY h:mm:ss
<b>%time02</b>	D.MM.YYYY h:mm
<b>%time03</b>	D.MM.YY h:mm
<b>%time04</b>	YYYY-MM-DD hh:mm:ss
<b>%time05</b>	YYYY-MM-DD hh:mm
<b>%time06</b>	YYYY-MM-DD h:mm
<b>%time07</b>	YY-MM-DD h:mm
<b>%time08</b>	DD.MM.YYYY
<b>%time09</b>	D.MM.YYYY
<b>%time10</b>	D.MM.YY
<b>%time11</b>	YYYY-MM-DD
<b>%time12</b>	YY-MM-DD
<b>%time13</b>	hh:mm:ss
<b>%time14</b>	h:mm:ss
<b>%time15</b>	h:mm
<b>%time99</b>	ISO 8601 calendar week



Properties:

- It comprises seven days
- It begins with Monday
- It is numbered sequentially
- The first calendar week (week 01) is the week with the first Thursday of the Gregorian year.

## 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 **SYSSTR ID10010 NR1/2** function)

The following formatting possibilities are available:

Input	Meaning	Example
<b>%main0</b>	Full path of the active NC program	<b>TNC:\MILL.h</b>
<b>%main1</b>	Path to the directory of the active NC program	<b>TNC:\</b>
<b>%main2</b>	Name of the active NC program	<b>MILL</b>
<b>%main3</b>	File type of the active NC program	<b>.H</b>
<b>%prog0</b>	Full path of the called NC program	<b>TNC:\HOUSE.h</b>
<b>%prog1</b>	Path to the directory of the called NC program	<b>TNC:\</b>
<b>%prog2</b>	Name of the called NC program	<b>HOUSE</b>
<b>%prog3</b>	File type of the active NC program	<b>.H</b>

## Engraving the counter reading

Cycle **225** allows you to engrave the current counter reading (provided in the MOD menu).

To do so, program Cycle **225** as usual and enter the text to be engraved, for example: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following:  
000000003

**Further information:** User's Manual for **Klartext Programming** or **ISO Programming**

### Operating notes

- In Test Run operating mode, the control simulates only the counter reading that you have 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 Cycle 232 FACE MILLING (option 19)

### ISO programming

#### G232

### Application



This function must be enabled and adapted by the machine manufacturer.

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:

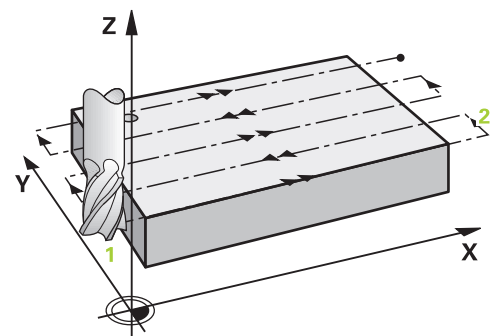
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1:** Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

### Cycle sequence

- 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.

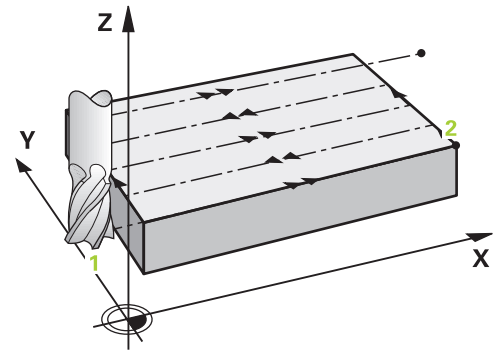
### Strategy Q389=0

- 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.

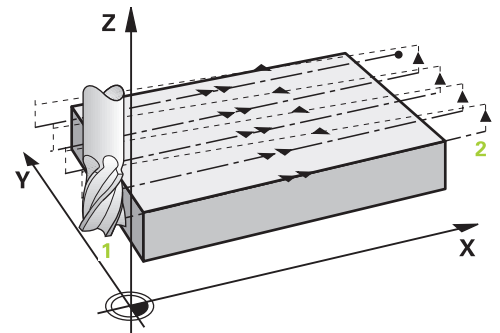


**Strategy Q389=1**

- 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.

**Notes**

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

**Notes on programming**

- 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.

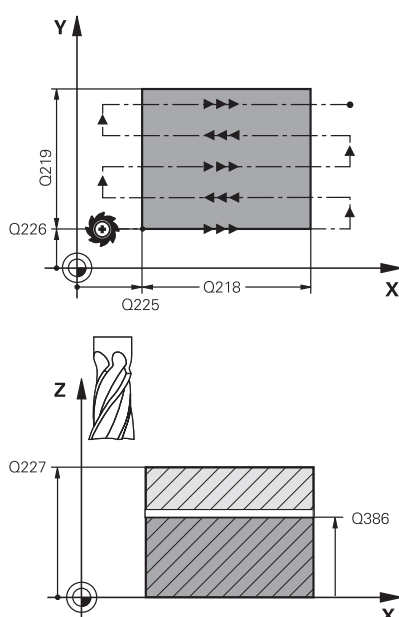


Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.



## Cycle parameters

### Help graphic



### Parameter

#### Q389 Machining strategy (0/1/2)?

Define how the control will machine the surface:

**0:** Meander machining, stepover at positioning feed rate outside the surface to be machined

**1:** Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined

**2:** Line-by-line machining, retraction and stepover at the positioning feed rate

Input: **0, 1, 2**

#### Q225 Starting point in 1st axis?

Define the starting point coordinate of the surface to be machined in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q226 Starting point in 2nd axis?

Define the starting point coordinate of the surface to be machined in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q227 Starting point in 3rd axis?

Coordinate of the workpiece surface used to calculate the infeeds. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q386 End point in 3rd axis?

Coordinate in the spindle axis on which the surface will be face-milled. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

#### Q218 First side length?

Length of the surface to be machined in the main axis of the working plane. Use the algebraic sign to specify the direction of the first milling path referenced to the **starting point in the 1st axis**. This value has an incremental effect.

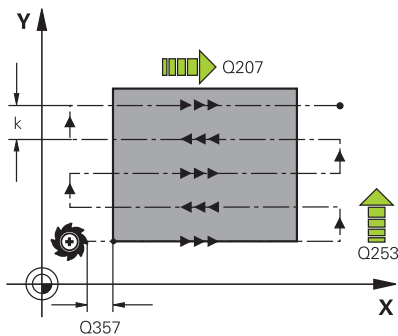
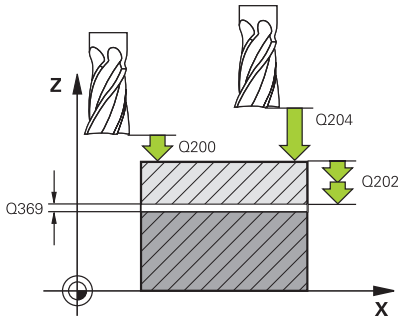
Input: **-99999.9999...+99999.9999**

#### Q219 Second side length?

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 referenced to the **STARTNG PNT 2ND AXIS**. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

## Help graphic



## Parameter

**Q202 Maximum plunging depth?**

**Maximum** infeed per cut. The control calculates the actual plunging depth from the difference between the end point and starting point in the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. This value has an incremental effect.

Input: **0...99999.9999**

**Q369 Finishing allowance for floor?**

Value used for the last infeed. This value has an incremental effect.

Input: **0...99999.9999**

**Q370 Max. path overlap factor?**

Maximum stepover factor k. The control calculates the actual stepover from the second side length (**Q219**) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. cutter radius when using a face-milling cutter), the control reduces the stepover accordingly.

Input: **0.001...1.999**

**Q207 Feed rate for milling?**

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

**Q385 Finishing feed rate?**

Traversing speed of the tool in mm/min while milling the last infeed

Input: **0...99999.999** or **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: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

**Q200 Set-up clearance?**

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. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

## Help graphic

## Parameter

**Q357 Safety clearance to the side?**

Parameter **Q357** influences the following situations:

**Approaching the first infeed depth:** **Q357** is the lateral distance from the tool to the workpiece.

**Roughing with the Q389 = 0 to 3 roughing strategies:** The surface to be machined is extended in **Q350 MILLING DIRECTION** by the value from **Q357** if no limit has been set in that direction.

**Side finishing:** The paths are extended by **Q357** in the **Q350 MILLING DIRECTION**.

Input: **0...99999.9999**

**Q204 2nd set-up clearance?**

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

## Example

11 CYCL DEF 232 FACE MILLING ~	
Q389=+2	;STRATEGY ~
Q225=+0	;STARTNG PNT 1ST AXIS ~
Q226=+0	;STARTNG PNT 2ND AXIS ~
Q227=+2.5	;STARTNG PNT 3RD AXIS ~
Q386=0	;END POINT 3RD AXIS ~
Q218=+150	;FIRST SIDE LENGTH ~
Q219=+75	;2ND SIDE LENGTH ~
Q202=+5	;MAX. PLUNGING DEPTH ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q370=+1	;MAX. OVERLAP ~
Q207=+500	;FEED RATE MILLING ~
Q385=+500	;FINISHING FEED RATE ~
Q253=+750	;F PRE-POSITIONING ~
Q200=+2	;SET-UP CLEARANCE ~
Q357=+2	;CLEARANCE TO SIDE ~
Q204=+50	;2ND SET-UP CLEARANCE

## 13.8 Cycle 238 MEASURE MACHINE STATUS (option 155)

### ISO programming

#### G238

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

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 sequence



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 On 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.

## Notes

### NOTICE

#### Danger of collision!

This cycle may perform extensive movements in one or more axes at 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 manufacturer 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.
- If, during a measurement, you set, for example, the feed rate potentiometer to zero, then the control will abort the cycle and display a warning. You can acknowledge the warning by pressing the **CE** key and then press the **NC start** key to run the cycle again.

## Cycle parameters

Help graphic	Parameter
	<b>Q570 Mode (0=test/1=measure)?</b> Define whether the control will perform a measurement of the machine status in test mode or in measurement mode: <b>0:</b> No measured data will be generated. You can control the axis movements with the feed rate and rapid traverse potentiometers <b>1:</b> This mode will generate measured data. You <b>cannot</b> control the axis movements with the feed rate and rapid traverse potentiometers Input: <b>0, 1</b>

### Example

```
11 CYCL DEF 238 MEASURE MACHINE STATUS ~
```

```
Q570=+0 ;MODE
```

## 13.9 Cycle 239 ASCERTAIN THE LOAD (option 143)

### ISO programming

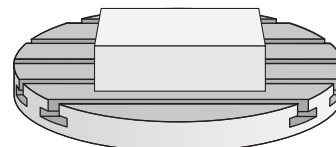
G239

### Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



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 sequence****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.



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 run will be performed.

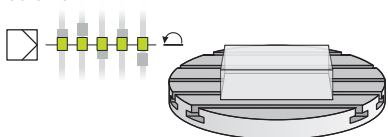
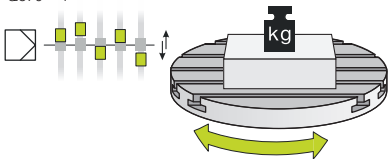
**Notes****NOTICE****Danger of collision!**

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

Help graphic	Parameter
<p>Q570 = 0</p>  <p>Q570 = 1</p> 	<p><b>Q570 Load (0 = Delete/1 = Ascertain)?</b></p> <p>Define whether the control will perform a LAC (Load Adaptive Control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters will be reset:</p> <p><b>0:</b> Reset LAC; the values most recently ascertained by the control are reset, and the control uses load-independent feedforward and controller parameters</p> <p><b>1:</b> 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.</p> <p>Input: <b>0, 1</b></p>

### Example

11 CYCL DEF 239 ASCERTAIN THE LOAD ~
Q570=+0 ;LOAD ASCERTATION



## 13.10 Cycle 18 THREAD CUTTING

### ISO programming

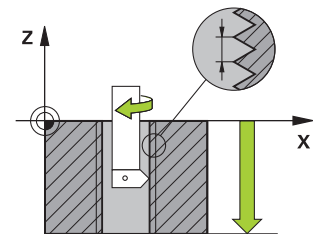
G86

### Application



This function must be enabled and adapted by the machine manufacturer.

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.



### Notes

#### 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.

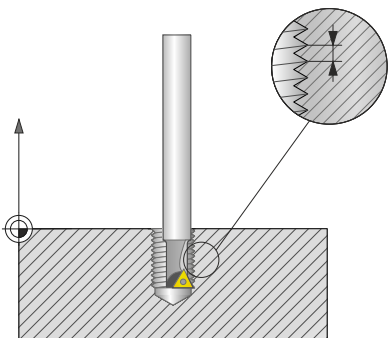
### Notes on programming

- Before calling this cycle, 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.

**Note regarding machine parameters**

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (spindle speed override is not active); the control then adjusts the spindle speed as required
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
  - **limitSpindleSpeed** (no. 113604): Spindle speed limit  
**True:** At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time.  
**False:** Limiting not active

**Cycle parameters**

Help graphic	Parameter
	<b>Total hole depth?</b> Enter the thread depth relative to the current position. This value has an incremental effect. Input: <b>-999999999...+999999999</b>
	<b>Thread pitch?</b> Enter the thread pitch. 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) Input: <b>-99.9999...+99.9999</b>

**Example**

11 CYCL DEF 18.0 THREAD CUTTING
12 CYCL DEF 18.1 DEPTH-20
13 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	■		223
8	MIRRORING	■		226
9	DWELL TIME	■		407
10	ROTATION	■		227
11	SCALING FACTOR	■		229
12	PGM CALL	■		408
13	ORIENTATION	■		410
14	CONTOUR	■		263
18	THREAD CUTTING		■	433
19	WORKING PLANE	■		231
20	CONTOUR DATA	■		267
21	PILOT DRILLING		■	270
22	ROUGH-OUT		■	272
23	FLOOR FINISHING		■	277
24	SIDE FINISHING		■	280
25	CONTOUR TRAIN		■	285
26	AXIS-SPECIFIC SCALING	■		230
27	CYLINDER SURFACE		■	371
28	CYLINDER SURFACE		■	374
29	CYL SURFACE RIDGE		■	379
32	TOLERANCE	■		411
39	CYL. SURFACE CONTOUR		■	383
200	DRILLING		■	70
201	REAMING		■	74
202	BORING		■	76
203	UNIVERSAL DRILLING		■	80
204	BACK BORING		■	85
205	UNIVERSAL PECKING		■	89
206	TAPPING		■	121
207	RIGID TAPPING		■	124

Cycle number	Cycle name	DEF active	CALL active	Page
208	BORE MILLING		■	97
209	TAPPING W/ CHIP BRKG		■	128
220	POLAR PATTERN	■		244
221	CARTESIAN PATTERN	■		247
224	DATAMATRIX CODE PATTERN	■		251
225	ENGRAVING		■	415
232	FACE MILLING		■	422
233	FACE MILLING (milling direction can be selected, take the side walls into account)		■	206
238	MEASURE MACHINE STATUS	■		428
239	ASCERTAIN THE LOAD	■		430
240	CENTERING		■	112
241	SINGLE-LIP D.H.DRLNG		■	102
247	PRESETTING	■		237
251	RECTANGULAR POCKET		■	163
252	CIRCULAR POCKET		■	170
253	SLOT MILLING		■	177
254	CIRCULAR SLOT		■	183
256	RECTANGULAR STUD		■	189
257	CIRCULAR STUD		■	195
258	POLYGON STUD		■	200
262	THREAD MILLING		■	135
263	THREAD MLLNG/CNTSNKG		■	139
264	THREAD DRILLNG/MLLNG		■	144
265	HEL. THREAD DRLG/MLG		■	149
267	OUTSIDE THREAD MLLNG		■	153
270	CONTOUR TRAIN DATA		■	283
271	OCM CONTOUR DATA		■	312
272	OCM ROUGHING		■	315
273	OCM FINISHING FLOOR		■	330
274	OCM FINISHING SIDE		■	334
275	TROCHOIDAL SLOT		■	289
276	THREE-D CONT. TRAIN		■	295
277	OCM CHAMFERING		■	337
1271	OCM RECTANGLE	■		343
1272	OCM CIRCLE	■		346
1273	OCM SLOT / RIDGE	■		349

Cycle number	Cycle name	DEF active	CALL active	Page
1278	OCM POLYGON	■		352
1281	OCM RECTANGLE BOUNDARY	■		355
1282	OCM CIRCLE BOUNDARY	■		357

## Index

### A

About this manual.....	22
Ascertain the load.....	430

### C

Contour cycles.....	260
Coordinate transformation	
Datum shift.....	223
Mirroring.....	226
Rotation.....	227
Scaling factor.....	229
Scaling factor, axis-specific...	230
Cycle.....	40
Calling.....	43
Define.....	41
Cycles and point tables.....	64
Cylinder surface cycles	
Fundamentals.....	370
Cylindrical surface cycles	
Contour.....	383
Cylindrical surface.....	371
Ridge.....	379
Slot.....	374

### D

Datum shift	
Programming.....	223
Drilling cycle	
Centering.....	112
Drilling Cycles.....	68
Back boring.....	85
Bore milling.....	97
Boring.....	76
Drilling.....	70
Reaming.....	74
Single-lip deep hole drilling....	102
Universal drilling.....	80
Universal pecking.....	89
Dwell time.....	407

### E

Engraving.....	415
----------------	-----

### F

Face milling.....	206, 422
Feature content level.....	28

### G

GLOBAL DEF.....	47
-----------------	----

### M

Machining patterns.....	54
Measure machine status.....	428

### O

OCM	
-----	--

Chamfering.....	337
Contour data.....	312
Cutting data calculator.....	321
Floor finishing.....	330
Roughing.....	315
Side finishing.....	334
Standard shapes.....	341
OCM cycles.....	308
With complex contour formula....	390
With simple contour formula.	400
OCM figures	
Circle.....	346
Circle boundary.....	357
Polygon.....	352
Rectangle.....	343
Rectangle boundary.....	355
Slot/ridge.....	349
Option.....	25

### P

Pattern	
Circle.....	244
DataMatrix Code.....	251
Lines.....	247
PATTERN DEF	
entering.....	55
using.....	55
Pattern definition with PATTERN	
DEF.....	54
frames.....	60
full circle.....	62
patterns.....	58
pitch circle.....	63
Point.....	56
Pecking.....	89
Pocket milling cycles	
Circular pocket.....	170
Rectangular pocket.....	163
Point patterns.....	242
Point tables with cycles.....	64
Presets, setting.....	237
Program call.....	408
Via cycle.....	408

### S

SL Cycles.....	260
3-D contour train.....	295
Contour.....	263
Contour data.....	267
Contour train.....	285
Contour train data.....	283
Floor finishing.....	277
Fundamentals.....	260
OCM chamfering.....	337
OCM contour data.....	312
OCM floor finishing.....	330
OCM fundamentals.....	308

OCM roughing.....	315
OCM side finishing.....	334
Pilot drilling.....	270
Roughing.....	272
Side finishing.....	280
Superimposed contours	264, 395
Trochoidal milling of contour	
slot.....	289
With complex contour formula....	390
With simple contour formula.	400
Slot milling cycles	
Circular slot.....	183
Slot milling.....	177
Software option.....	25
Spindle orientation.....	410
Stud milling cycles	
Circular stud.....	195
Polygon stud.....	200
Rectangular stud.....	189

### T

Table of cycles.....	436
Machining cycles.....	436
Tapping.....	120
With chip breaking.....	128
With floating tap holder.....	121
Without floating tap holder....	124
Thread cutting.....	433
Thread milling	
Fundamentals.....	133
Helical thread drilling/milling.	149
Inside.....	135
Outside.....	153
Thread drilling/milling.....	144
Thread milling/countersinking....	139
Tilt working plane	
Procedure.....	236
Tolerance.....	411

### W

Working plane.....	231
--------------------	-----

# 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](mailto:info@heidenhain.de)

**Technical support** FAX +49 8669 32-1000

**Measuring systems** ☎ +49 8669 31-3104

E-mail: [service.ms-support@heidenhain.de](mailto:service.ms-support@heidenhain.de)

**NC support** ☎ +49 8669 31-3101

E-mail: [service.nc-support@heidenhain.de](mailto:service.nc-support@heidenhain.de)

**NC programming** ☎ +49 8669 31-3103

E-mail: [service.nc-pgm@heidenhain.de](mailto:service.nc-pgm@heidenhain.de)

**PLC programming** ☎ +49 8669 31-3102

E-mail: [service.plc@heidenhain.de](mailto:service.plc@heidenhain.de)

**APP programming** ☎ +49 8669 31-3106

E-mail: [service.app@heidenhain.de](mailto:service.app@heidenhain.de)

[www.heidenhain.de](http://www.heidenhain.de)

[www.klartext-portal.com](http://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.

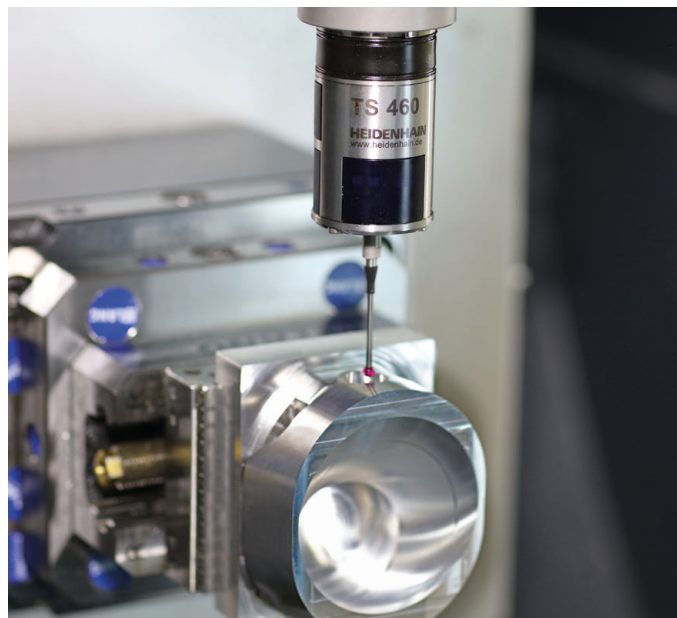
### Workpiece touch probes

**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

