HEIDENHAIN

TNC7 basic

User's Manual
for Machining Cycles

NC Software
81762x-20

English (en)
10/2025

Table of contents

1 New and modified functions 21
2 About the User's Manual 25
3 About the product 39
4 First steps 57
5 NC and programming fundamentals 83
6 Programming techniques 97
7 Contour and point definitions 101
8 Cycles for drilling, centering, and thread machining 175
9 Milling cycles 259
10 Coordinate transformation 415
11 Control functions 429
12 Component monitoring (#155 / #5-02-1) 441
13 Multiple-axis machining 445
14 Programming with variables 465
15 User aids 473

1 New and modified functions..... 21

1.1 New functions..... 23

1.1.1 Process Tracking Interface (#3-04-1*)..... 23
1.1.2 Dynamic Collision Monitoring DCM (#140 / #5-03-2)..... 23

1.2 Modified or extended functions..... 24

1.2.1 Milling cycles..... 24

2 About the User's Manual 25

2.1 Target group: Users 26

2.2 Available user documentation 27

2.3 Types of notes used 29

2.4 Notes on using NC programs 31

2.5 User's Manual as integrated product aid: TNCguide 32

2.5.1 Searching in TNCguide 35

2.5.2 Copying NC examples to clipboard 36

2.6 Contacting the editorial staff 37

3 About the product 39

3.1 The TNC7 basic 40

3.1.1 Proper and intended use 40
3.1.2 Data Architecture and Access Options 41
3.1.3 Intended place of operation 42

3.2 Safety precautions 43

3.3 Software 45

3.3.1 Software options 46
3.3.2 Information on licensing and use 52

3.4 Areas of the control's user interface 53

3.5 Overview of the operating modes 55

4 First steps..... 57

4.1 Programming and simulating a workpiece..... 58

4.1.1 Programming the workpiece..... 58
4.1.2 Simulating a workpiece..... 79

5 NC and programming fundamentals 83

5.1 Working with cycles 84

5.1.1 General information on cycles 84
5.1.2 General information about touch probe cycles 93
5.1.3 Machine-specific cycles 94
5.1.4 Available cycle groups 95

6 Programming techniques..... 97

6.1 Cycle 12 PGM CALL..... 98

6.1.1 Cycle parameters..... 99

7 Contour and point definitions 101

7.1 Superimposing contours 102
7.1.1 Fundamentals 102
7.1.2 Subprograms: overlapping pockets 102
7.1.3 Surface resulting from sum 103
7.1.4 Surface resulting from difference 104
7.1.5 Surface resulting from intersection 105

7.2 Cycle 14 CONTOUR 106
7.2.1 Cycle parameters 106

7.3 Simple contour formula 107
7.3.1 Fundamentals 107
7.3.2 Entering a simple contour formula 109
7.3.3 Machining contours with SL or OCM cycles 110

7.4 Complex contour formula 111
7.4.1 Fundamentals 111
7.4.2 Selecting an NC program with contour definition 114
7.4.3 Defining a contour description 115
7.4.4 Entering a complex contour formula 116
7.4.5 Superimposed contours 117
7.4.6 Machining contours with SL or OCM cycles 119

7.5 Point tables 120
7.5.1 Selecting the point table in the NC program with **SEL PATTERN** 121
7.5.2 Calling the cycle with a point table 121

7.6 Pattern definition with PATTERN DEF 123
7.6.1 Defining individual machining positions 125
7.6.2 Defining a single row 126
7.6.3 Defining an individual pattern 127
7.6.4 Defining an individual frame 128
7.6.5 Defining a full circle 130
7.6.6 Defining a pitch circle 131
7.6.7 Example: Using cycles in conjunction with PATTERN DEF 132

7.7 Pattern definition cycles 134
7.7.1 Overview 134
7.7.2 Cycle 220 **POLAR PATTERN** 136
7.7.3 Cycle 221 **CARTESIAN PATTERN** 139
7.7.4 Cycle 224 **DATAMATRIX CODE PATTERN** 142
7.7.5 Programming examples 148

7.8 OCM cycles for figure definition 150
7.8.1 Overview 150
7.8.2 Fundamentals 150
7.8.3 Cycle 1271 **OCM RECTANGLE** (#167 / #1-02-1) 153
7.8.4 Cycle 1272 **OCM CIRCLE** (#167 / #1-02-1) 157
7.8.5 Cycle 1273 **OCM SLOT / RIDGE** (#167 / #1-02-1) 160
7.8.6 Cycle 1274 **OCM CIRCULAR SLOT** (#167 / #1-02-1) 163
7.8.7 Cycle 1278 **OCM POLYGON** (#167 / #1-02-1) 166
7.8.8 Cycle 1281 **OCM RECTANGLE BOUNDARY** (#167 / #1-02-1) 170
7.8.9 Cycle 1282 **OCM CIRCLE BOUNDARY** (#167 / #1-02-1) 172

8 Cycles for drilling, centering, and thread machining 175

8.1 Overview 176

8.2 Conditional stops in drilling and threading operations 178

8.3 Drilling 179

8.3.1 Cycle 200 **DRILLING** 179
8.3.2 Cycle 201 **REAMING** 183
8.3.3 Cycle 202 **REAMING** 185
8.3.4 Cycle 203 **UNIVERSAL DRILLING** 189
8.3.5 Cycle 205 **UNIVERSAL PECKING** 194
8.3.6 Cycle 208 **BORE MILLING** 201
8.3.7 Cycle 241 **SINGLE-LIP D.H.DRLNG** 206

8.4 Countersinking and centering 214

8.4.1 Cycle 204 **BACK BORING** 214
8.4.2 Cycle 240 **CENTERING** 218

8.5 Tapping 221

8.5.1 Cycle 18 **THREAD CUTTING** 221
8.5.2 Cycle 206 **TAPPING** 224
8.5.3 Cycle 207 **RIGID TAPPING** 227
8.5.4 Cycle 209 **TAPPING W/ CHIP BRKG** 231

8.6 Thread milling 235

8.6.1 Fundamentals of thread milling 235
8.6.2 Cycle 262 **THREAD MILLING** 237
8.6.3 Cycle 263 **THREAD MLLNG/CNTSNKG** 241
8.6.4 Cycle 264 **THREAD DRILLNG/MLLNG** 246
8.6.5 Cycle 265 **HEL. THREAD DRLG/MLG** 251
8.6.6 Cycle 267 **OUTSIDE THREAD MLLNG** 255

9 Milling cycles ... 259

9.1 Overview ... 260

9.2 Conditional stops in milling cycles ... 263

9.3 Milling pockets ... 264

9.3.1 Cycle 251 **RECTANGULAR POCKET** ... 264
9.3.2 Cycle 252 **CIRCULAR POCKET** ... 271
9.3.3 Cycle 253 **SLOT MILLING** ... 277
9.3.4 Cycle 254 **CIRCULAR SLOT** ... 283

9.4 Milling studs ... 290

9.4.1 Cycle 256 **RECTANGULAR STUD** ... 290
9.4.2 Cycle 257 **CIRCULAR STUD** ... 296
9.4.3 Cycle 258 **POLYGON STUD** ... 301
9.4.4 Programming examples ... 307

9.5 Milling contours with SL cycles ... 310

9.5.1 Fundamentals ... 310
9.5.2 Cycle 20 **CONTOUR DATA** ... 313
9.5.3 Cycle 21 **PILOT DRILLING** ... 315
9.5.4 Cycle 22 **ROUGH-OUT** ... 317
9.5.5 Cycle 23 **FLOOR FINISHING** ... 321
9.5.6 Cycle 24 **SIDE FINISHING** ... 324
9.5.7 Cycle 270 **CONTOUR TRAIN DATA** ... 328
9.5.8 Cycle 25 **CONTOUR TRAIN** ... 330
9.5.9 Cycle 275 **TROCHOIDAL SLOT** ... 335
9.5.10 Cycle 276 **THREE-D CONT. TRAIN** ... 341
9.5.11 Programming examples ... 346

9.6 Milling contours with OCM cycles (#167 / #1-02-1) ... 352

9.6.1 Fundamentals ... 352
9.6.2 Cycle 271 **OCM CONTOUR DATA** (#167 / #1-02-1) ... 360
9.6.3 Cycle 272 **OCM ROUGHING** (#167 / #1-02-1) ... 363
9.6.4 Cycle 273 **OCM FINISHING FLOOR** (#167 / #1-02-1) ... 368
9.6.5 Cycle 274 **OCM FINISHING SIDE** (#167 / #1-02-1) ... 371
9.6.6 Cycle 277 **OCM CHAMFERING** (#167 / #1-02-1) ... 374
9.6.7 Programming examples ... 378

9.7 Milling planes ... 390

9.7.1 Cycle 232 **FACE MILLING** ... 390
9.7.2 Cycle 233 **FACE MILLING** ... 397

9.8 Engraving ... 407

9.8.1 Cycle 225 **ENGRAVING** ... 407

10 Coordinate transformation 415

10.1 Coordinate transformation cycles 416

10.1.1 Fundamentals 416
10.1.2 Cycle 8 **MIRRORING** 417
10.1.3 Cycle 10 **ROTATION** 419
10.1.4 Cycle 11 **SCALING FACTOR** 421
10.1.5 Cycle 26 **AXIS-SPECIFIC SCALING** 423
10.1.6 Cycle **247 PRESETTING** 424
10.1.7 Example: Coordinate conversion cycles 426

11 Control functions 429

11.1 Cycles with control function 430

11.1.1 Overview 430
11.1.2 Conditional stops in cycles with control functions 430
11.1.3 Cycle 9 **DWELL TIME** 431
11.1.4 Cycle 13 **ORIENTATION** 432
11.1.5 Cycle 32 **TOLERANCE** 434
11.1.6 Cycle 239 **ASCERTAIN THE LOAD** (#143 / #2-22-1) 439

12 Component monitoring (#155 / #5-02-1)..... 441

12.1 Cycle 238 MEASURE MACHINE STATUS (#155 / #5-02-1)..... 442

12.1.1 Cycle parameters..... 443

13 Multiple-axis machining 445

13.1 Cycles for cylinder surface machining 446

13.1.1 Overview 446
13.1.2 Conditional stops in cylinder surface cycles 446
13.1.3 Cycle 27 **CYLINDER SURFACE** (#8 / #1-01-1) 447
13.1.4 Cycle 28 **CYLINDRICAL SURFACE SLOT** (#8 / #1-01-1) 450
13.1.5 Cycle 29 **CYL SURFACE RIDGE** (#8 / #1-01-1) 454
13.1.6 Cycle 39 **CYL. SURFACE CONTOUR** (#8 / #1-01-1) 458
13.1.7 Programming examples 461

14 Programming with variables 465

14.1 Program defaults for cycles 466

14.1.1 Overview 466
14.1.2 Entering GLOBAL DEF definitions 466
14.1.3 Using GLOBAL DEF information 467
14.1.4 Global data valid everywhere 468
14.1.5 Global data for drilling operations 469
14.1.6 Global data for milling operations with pocket cycles 470
14.1.7 Global data for milling operations with contour cycles 471
14.1.8 Global data for positioning behavior 471

15 User aids 473

15.1 OCM cutting data calculator (#167 / #1-02-1) 474

15.1.1 Fundamentals of the OCM cutting data calculator 474
15.1.2 Operation 475
15.1.3 Fillable form 476
15.1.4 Process parameters 481
15.1.5 Achieving an optimum result 481

1

New and modified functions

Available documentation

TNC7 basic complete edition

The split editions of the User's Manual contain only the new and modified functions that are relevant to the corresponding User's Manual. The **complete edition** contains all new and modified functions of this software version that are relevant to the user.

ID: 1411730-xx

You can download this documentation free of charge from the HEIDENHAIN website.

https://content.heidenhain.de/doku/tnc_guide/html/en/

Overview of new and modified software functions

The additional documentation **Overview of New and Modified Software Functions** contains all new and modified functions of the current and previous software versions that are relevant to the user.

ID: 1443541-xx

You can download this documentation free of charge from the HEIDENHAIN website.

TNCguide

1.1 New functions

1.1.1 Process Tracking Interface (#3-04-1*)

Topic	Description
Software option PTI 8 channels (#3-04-1*)	This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with **SIK2**. You can order this software option up to two times, each time selecting up to eight channels for recording.

1.1.2 Dynamic Collision Monitoring DCM (#140 / #5-03-2)

Topic	Description
The Collision Monitoring software option	The software option Collision Monitoring (#40 / #5-03-1) is no longer available. To be able to use the Dynamic Collision Monitoring (DCM) functions in the future, you need the software option Collision Monitoring v2 (#140 / #5-03-2) If the software option Collision Monitoring v2 (#140 / #5-03-2) is not available after a software update, the control will display a warning.

1.2 Modified or extended functions

1.2.1 Milling cycles

Topic	Description
Q255 DWELL TIME	In Cycle **204 BACK BORING**, the input value of this parameter was extended from zero to four decimal places. **Further information:** "Cycle 204 BACK BORING ", Page 214

2

About the User's Manual

2.1 Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:

- Operating the machine
  - Setting up tools
  - Setting up workpieces
  - Machining workpieces
  - Eliminating possible errors during program run
- Creating and testing NC programs
  - Creating NC programs at the control or externally using a CAM system
  - Using the Simulation mode to test the NC programs
  - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:

- Basic technical understanding (e.g., spatial imagination and the ability to read technical drawings)
- Basic knowledge in the field of metal cutting (e.g., understanding the meaning of material-specific parameters)
- Safety instructions (e.g., understanding possible dangers and how to avoid them)
- Training on the machine (e.g., comprehending axis directions and the machine configuration)

HEIDENHAIN offers separate information products for other target groups:

- Leaflets and overview of the product portfolio for potential buyers
- Service Manual for service technicians
- Technical Manual for machine manufacturers

Additionally, HEIDENHAIN provides users and lateral entrants with a wide range of training opportunities in the field of NC programming.
HEIDENHAIN training portal

In line with the target group, this User's Manual only contains information on the operation and use of the control. The information products for other target groups contain information on further product life phases.

2.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as a User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator's manual and operating instructions.

The User's Manual for the control is available in the variants below:

- As a PDF file containing a **complete edition** of the User's Manual with all contents

  ID: 1411730-xx

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- As an HTML file with all contents for use as the integrated product aid **TNCguide**: directly on the control

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- As a printed version, sub-divided into the modules below:
  - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for executing NC programs.

    ID: 1410286-xx
  - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe cycles and machining cycles are not included.

    ID: 1409856-xx
  - The **Machining Cycles** User's Manual contains all machining cycles functions.

    ID: 1410289-xx
  - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions related to the touch-probe cycles.

    ID: 1410290-xx

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended use", Page 40

Complete edition and TNCguide for the TNC7 basic

Binding of the printed version of the User's Manual is only possible if a certain number of pages is not exceeded.

Die **complete edition** and **TNCguide** may include additional sample programs for NC functions that are not contained in the printed version.

You can download this documentation free of charge from the HEIDENHAIN website.

Further information products for users

The following information products are available:

- The **Overview of New and Modified Software Functions** informs you about the innovations of specific software versions.

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- **Overview of the machine parameters, error numbers and system data**, providing the following functions:
  - Machine parameters of the **MPs for setters** application
  - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
  - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- **HEIDENHAIN brochures** inform you about products and services from HEIDENHAIN (e.g., software options of the control).

  HEIDENHAIN brochures
- The **NC Solutions** database offers solutions for frequently occurring tasks.

  HEIDENHAIN NC solutions

2.3 Types of notes used

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

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

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

The information symbol indicates a **tip**.
A tip provides important 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 indicates a **cross reference**.
A cross reference leads to external documentation, for example the documentation of your machine manufacturer or other supplier.

2.4 Notes on using NC programs

NC programs contained in this User's Manual are suggestions for solutions. The NC programs or individual NC blocks must be adapted before being used on a machine.

Change the following contents as needed:

- Tools
- Cutting parameters
- Feed rates
- Clearance height or safe position
- Machine-specific positions (e.g., with **M91**)
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

In addition, test the NC programs using the simulation before the actual program run.

With a program test you determine whether the NC program can be used with the available software options, the active machine kinematics and the current machine configuration.

2.5 User's Manual as integrated product aid: TNCguide

Application

The integrated product aid **TNCguide** offers the full content of all User's Manuals.

Further information: "Available user documentation", Page 27

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended use", Page 40

Related topics

- The **Help** workspace
  Further information: Programming and Testing User's Manual

Requirement

In the factory default setting, the control offers the integrated product aid **TNCguide** in German and English language versions.

If the control cannot find a **TNCguide** language version matching the selected dialog language, it opens **TNCguide** in English.

If the control cannot find a **TNCguide** language version, it opens an information page with instructions. With the link available there and the steps provided, you can supplement the files missing in the control.

> You can also open the information page manually by selecting the **index.html** file (for example, at **TNC:\tncguide\en\readme**). The path depends on the desired language version (e.g., **en** for English).
>
> With the steps provided you can also update the **TNCguide** version. Updating may be required (e.g., after a software update).

Description of function

The integrated product aid **TNCguide** can be selected within the **Help** application or in the **Help** workspace.

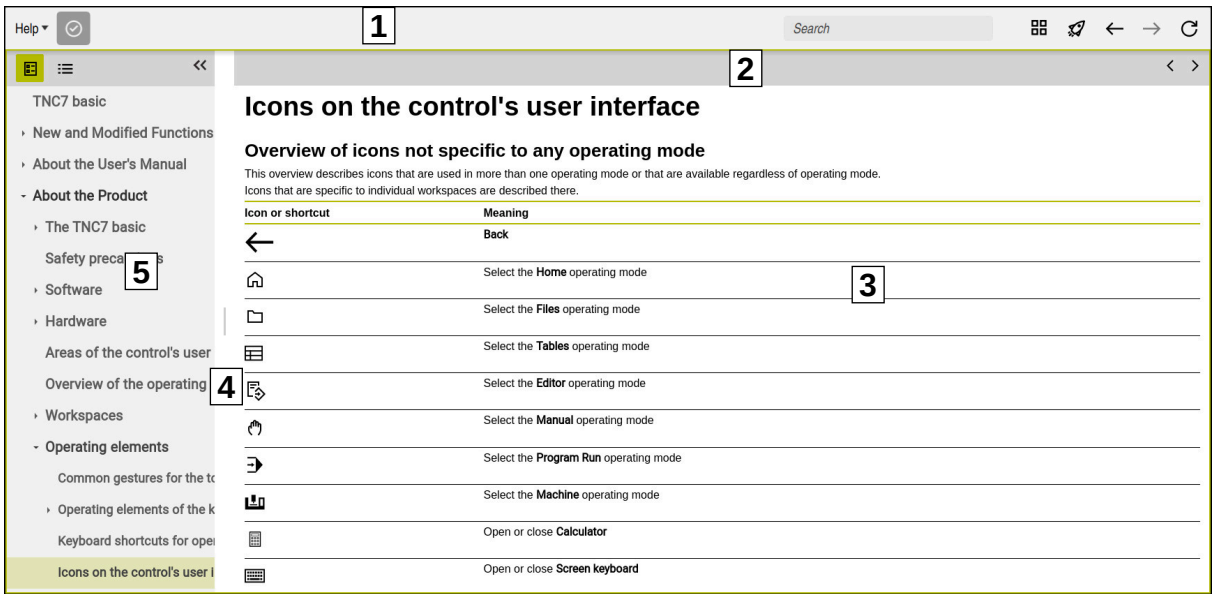
Further information: "The Help application", Page 33

Further information: Programming and Testing User's Manual

Operation of **TNCguide** is identical in both cases.

Further information: "Icons", Page 34

The Help application



Open **TNCguide** in the **Help** workspace

TNCguide includes the following areas:

1 Title bar of the **Help** workspace
Further information: "The Help workspace", Page 34

2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 34

3 Content column of the **TNCguide**

4 Separator between the columns of the **TNCguide**
Adjust the column width by means of the separator.

5 Navigation column of the **TNCguide**

Icons

The Help workspace

The **Help** workspace within the **Help** application includes the following icons:

Icon	Meaning
	Open or close the **Search results** column **Further information:** "Searching in TNCguide", Page 35
	Open Home page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., **TNCguide**). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function.
	Open Tutorials
	Navigate Navigate between the contents opened recently
	Refresh

TNCguide

The integrated **TNCguide** product aid includes the following icons:

Icon	Meaning
	Open Structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Open Index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
	Navigate Display previous or next page within the documentation
	Open or close Display or hide the navigation
	Copy Copy NC examples to the clipboard **Further information:** "Copying NC examples to clipboard", Page 36

Context-sensitive help

You can open **TNCguide** for the current context. Context-sensitive help means that the relevant information is displayed directly (e.g., for the selected item or the current NC function).

To call context-sensitive help, the following elements are available:

Icon or key	Meaning
?	**Help** icon If you select the icon and then one of the items in the user interface, the control will open the associated information in **TNCguide**.
HELP	**HELP** key If you press the **HELP** key while editing an NC block, the control will display the associated information in **TNCguide**.

If you call TNCguide in a certain context, the control opens the contents in a pop-up window. If you select the **Show more** button, the control will open **TNCguide** in the **Help** application.

Further information: "The Help application", Page 33

If the **Help** workspace is already open, the control displays **TNCguide** there and will not open a pop-up window.

Further information: Programming and Testing User's Manual

2.5.1 Searching in TNCguide

Using the search function, you can search for the entered search terms within the open documentation.

Use the search function as follows:

- Enter a string in **Search**

> The search starts automatically after you enter a character.
> If you wish to delete the entry, use the X symbol within the entry field.

> The control opens the column containing the search results.
> The control marks references also within open content pages.
- Select the reference
> The control opens the selected content.
> The control continues displaying the results of the last search.
- Select an alternative reference if necessary
- Enter a new character string if required

2.5.2 Copying NC examples to clipboard

Use the copy function to copy NC examples from the documentation to the NC editor.

To use the copy function:

- Navigate to the desired NC example
- Expand **Notes on using NC programs**
- Read and follow **Notes on using NC programs**

  Further information: "Notes on using NC programs", Page 31

- Copy NC example to clipboard

> The button switches colors while copying.

> The clipboard contains the entire content of the copied NC example.

- Insert the NC example into the NC program
- Adapt the inserted content according to the **Notes on using NC programs**
- Use the Simulation mode to test the NC program

  Further information: Programming and Testing User's Manual

2.6 Contacting the editorial staff

Would you like any changes, or have you found any errors?

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

tnc-userdoc@heidenhain.com

3

About the product

3.1 The TNC7 basic

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 basic additionally offers you form-based or graphical programming to reach the desired result quickly and easily.

Software options and optional hardware extensions can be used for flexibly increasing the range of functions and ease of use.

Operation is made easier, for example, by using touch probes, handwheels or a 3D mouse.

Further information: User's Manual for Setup and Program Run

Definitions

Abbreviation	Definition
TNC	**TNC** is derived from the acronym **CNC** (computerized numerical control). The **T** (tip or touch) stands for the capability of entering NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.
basic	The addition basic indicates that the control provides all required basic functions for universal milling or drilling.

3.1.1 Proper and intended use

The information about proper and intended use supports you in safely handling a product such as a machine tool.

The control is a machine component but not a complete machine. This User's Manual describes the use of the control. Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

HEIDENHAIN sells controls designed for milling and turning machines as well as for machining centers with up to 24 axes. If you as a user face a different constellation, then contact the owner immediately.

HEIDENHAIN contributes additionally to enhancing your safety and that of your products, notably by taking customer feedback into consideration. This results, for example, in function adaptations of the controls and safety precautions in the information products.

Contribute actively to increasing the operational safety by reporting any missing or misleading information.

Further information: "Contacting the editorial staff", Page 37

3.1.2 Data Architecture and Access Options

Directories

While in use, the control will store various types of data on its internal data carrier.

Partition or directory	Data
root	Operating system
/mnt/tnc/	Process, application, and user data Examples: NC programs, tables or 3D models
/mnt/plc/	Machine manufacturer data Examples: PLC program, machine configuration or additional software
/mnt/sys/	NC software

The type, format, and amount of data stored depend on various factors. For example, the software options that are available and actually used as well as the functions used, including the current settings, have a great influence here.

Data acquisition

While in use, the control will generate different pieces of data. Depending on the functions in use, for example, data acquisition is performed continuously and in real time.

Data acquisition examples:

- Backup

  You can manually create a backup copy of the TNC partition in order to avoid potential loss of data.
- Service file

  In the event of a fault, you can manually create a service file that supports troubleshooting by a service technician.
- User administration

  If user administration is active, the control assigns defined roles and permissions to the logged-on users. When a user logs on, the control will acquire personal or anonymous data in connection with its use.
- Tool-usage file

  Using tool-usage files, the control can, for example, check whether the tool's remaining life is sufficient for the current NC program.
- Log functions or log files (e.g., from touch-probe cycles)

  Using various log files, you can, for example, check and compare the results of touch-probe cycles.

  The control needs other function-dependent log files to, for example, detect process variations.

For more details on data acquisition, please refer to the sections describing the individual functions and the **Settings** application.

Data transfer and data access

Machine manufacturers, machine operators, or users can transfer data to external data carriers or drives, for example, and set up and configure a data transfer functionality. The user has control over and responsibility for the stored data (e.g., with respect to safety and the duration of storage).

Examples of data transfer:

- **HEIDENHAIN DNC** (#18 / #3-03-1)

  This software option enables external Windows applications to access data of the control via the TCP/IP protocol.
- **OPC UA NC Server** (#56-61 / #3-02-1*)

  These software options include OPC UA, a standardized interface that allows external access to the data and functions of your control.
- USB devices

  USB devices can be used to transfer data or make external backups.

> For more details on data transfer, please refer to the sections describing the individual functions and the **Settings** application.

Data access via file management:

- Drives, folders, and files

  You can access external data or make external backups by using connected drives or USB devices. For example, you can manage, create, edit, or delete folders and files.
- Functions and settings

  In the file manager you can search for certain contents or display hidden or dependent files.

> For further details on file access, please refer to the section on the **Files** operating mode.
>
> Please observe the safety instructions and information given by the machine manufacturer in order to avoid loss of data or jeopardize the functionality of your system.

3.1.3 Intended place of operation

In accordance with the DIN EN 50370-1 standard for electromagnetic compatibility (EMC), the control is approved for use in industrial environments.

Definitions

Guideline	Definition
DIN EN 50370-1:2006-02	This standard deals, among other things, with interference emissions and immunity to interference of machine tools.

3.2 Safety precautions

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

The following safety precautions refer exclusively to the control as an individual component but not to the specific complete product, i.e. the machine tool.

Refer to your machine manual.

Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

The following overview contains exclusively the generally valid safety precautions. Pay attention to additional safety precautions that may vary with the configuration and are given in the following chapters.

For ensuring maximum safety, all safety precautions are repeated at the relevant places within the chapters.

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- Devices should be connected or removed only by authorized service technicians
- Only switch on the machine via a connected handwheel or a secured connection

DANGER

Caution: hazard to the user!

Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- Read and follow the machine manual
- Read and follow the safety precautions and safety symbols
- Use the safety devices

WARNING

Caution: hazard to the user!

Manipulated data records or software can lead to an unexpected behavior of the machine. Malicious software (viruses, trojans, malware, or worms) can cause changes to data records and software.

- Check any removable memory media for malicious software before using them
- Start the internal web browser only from within the sandbox

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.

- Pay attention to the information on the screen
- If necessary, move to a safe position before referencing the axes
- Watch out for possible collisions

NOTICE

Danger of collision!

The control uses the defined tool length from the tool table for compensating for the tool length. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform tool length compensation or a collision check for tools with a length of **0** and after a **TOOL CALL 0**. There is a risk of collision during subsequent tool positioning movements!

- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

NOTICE

Danger of collision!

NC programs that were created on older controls can lead to unexpected axis movements or error messages on current control models. Danger of collision during machining!

- Check the NC program or program section using the graphic simulation
- Carefully test the NC program or program section in **Program Run** Single Block mode

NOTICE

Caution: Data may be lost!

If you do not properly remove a connected USB device during a data transfer, then data may be damaged or deleted!

- Use the USB port only for transferring or backing up data do not use it for editing and executing NC programs
- Use the **Eject** soft key to remove a USB device when data transfer is complete

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss regardless of the control's status!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

3.3 Software

This User's Manual describes the functions for setting up the machine as well as for programming and running your NC programs. These functions are available for a control featuring the full range of functions.

The actual range of functions depends, among other things, on the enabled software options.
Further information: "Software options", Page 46

The table shows the NC software numbers described in this User's Manual.

HEIDENHAIN has simplified the version schema, starting with NC software version 16:

- The publication period determines the version number.
- All control models of a publication period have the same version number.
- The version number of the programming stations corresponds to the version number of the NC software.

NC software number	Product
817620-20	TNC7 basic
817625-20	TNC7 basic Programming Station

Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Check, on the basis of the machine tool manual, whether the machine manufacturer has adapted the functions of the control.

If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

3.3.1 Software options

Software options define the range of functions of the control. The optional functions are either machine- or application-specific. The software options give you the possibility of adapting the control to your individual needs.

You can check which software options are enabled on your machine.

Further information: User's Manual for Setup and Program Run

The TNC7 basic features various software options that the machine manufacturer may enable separately, even at a later point in time. The following overview includes only those software options that are relevant for you.

The software options are saved on the **SIK** (System Identification Key) plug-in board. The TNC7 basic can be equipped with a **SIK** or **SIK2** plug-in board. Depending on which one is used, the numbers of the software options differ.

> The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions.
>
> The parentheses enclose the **SIK** and **SIK2** option numbers, separated by a slash, for example: (#18 / #3-03-1).
>
> The Technical Manual contains information about additional software options that are relevant to the machine manufacturer.

SIK2 definitions

SIK2 option numbers are structured by <class>-<option>-<version>:

Class	The function is effective for the following areas: ■ 1: Programming, simulation, and process setup ■ 2: Part quality and productivity ■ 3: Interfaces ■ 4: Technology functions and quality assessment ■ 5: Process stability and monitoring ■ 6: Machine configuration ■ 7: Developer tools
Option	Sequential number within each class
Version	New versions of software options are released if, for example, its features have been changed.

You can order some software options with **SIK2** more than once in order to obtain multiple variants of the same function (e.g., if you need to enable multiple control loops for the axes). In the User's Manual, these software option numbers are identified by an asterisk (*****).

The control indicates in the **SIK** menu item of the **Settings** application whether a software option has been enabled, and if so, how often. The control also shows whether is equipped with the **SIK** or **SIK2**.

Further information: User's Manual for Setup and Program Run

Overview

> Keep in mind that particular software options also require hardware extensions.
>
> **Further information:** User's Manual for Setup and Program Run

Software option	Definition and application
Control Loop Qty. (#0-3 / #6-01-1*)	**Additional control loop** A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required, for example, for detachable and motor-driven tilting tables. If your control features a **SIK2**, you can order this software option multiple times and enable up to 8 control loops.
Adv. Function Set 1 (#8 / #1-01-1)	**Advanced functions (set 1)** On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: ■ Tilting the working plane (e.g., with **PLANE SPATIAL**) **Further information:** Programming and Testing User's Manual ■ Programming of contours on an unrolled cylinder surface (e.g., with Cycle **27 CYLINDER SURFACE** **Further information:** "Cycle 27 CYLINDER SURFACE (#8 / #1-01-1)", Page 447 ■ Programming the rotary axis feed rate in mm/min with **M116** **Further information:** Programming and Testing User's Manual ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.
Adv. Function Set 2 (#9 / #4-01-1)	**Advanced functions (set 2)** On machines with rotary axes, this software option enables simultaneous 4-axis machining of workpieces. The software option includes the following functions: ■ **TCPM** (tool center point management): Automatic tracking of linear axes during rotary axis positioning **Further information:** Programming and Testing User's Manual ■ Running of NC programs with vectors, including optional 3D tool compensation **Further information:** Programming and Testing User's Manual ■ Manual moving of axes in the active tool coordinate system **T-CS**

Software option	Definition and application
Touch Probe Functions (#17 / #1-05-1)	**Touch-probe functions** This software option is used to program and execute automatic probing processes. If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled. The software option includes the following functions: ■ Automatic compensation of workpiece misalignment ■ Automatic setting of workpiece presets ■ Automatic measurement of workpieces ■ Automatic measurement of tools The touch-probe functions reduce the setup effort and increase accuracy when machining workpieces.
HEIDENHAIN DNC (#18 / #3-03-1)	**HEIDENHAIN DNC** This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Adv. Function Set 3 (#21 / #4-02-1)	**Advanced functions (set 3)** This software option offers additional ease of use with two powerful miscellaneous functions. The software option includes the following miscellaneous functions: ■ **M120** for machining small contour steps without error message and contour damage **Further information:** Programming and Testing User's Manual ■ **M118** for superimposed handwheel positioning during program run **Further information:** Programming and Testing User's Manual The advanced functions (set 3) reduce the setup effort and increase flexibility during program run.
CAD Import (#42 / #1-03-1)	**CAD Import** This software option is used to select positions and contours from CAD files and to transfer them into an NC program. With the CAD Import option you reduce the programming effort and prevent typical errors such as the incorrect entry of values. In addition, CAD Import contributes to paperless manufacturing. **Further information:** User's Manual for Setup and Program Run
Adaptive Feed Contr. (#45 / #2-31-1)	**Adaptive Feed Control (AFC)** This software option enables an automatic feed control that depends on the current spindle load. The control increases the feed rate as the load decreases and reduces the feed rate as the load increases. With AFC you can shorten machining times without adapting the NC program, while at the same time preventing machine damage from overload. **Further information:** User's Manual for Setup and Program Run

Software option	Definition and application
KinematicsOpt (#48 / #2-01-1)	**KinematicsOpt** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsOpt the control can correct position errors on rotary axes and thus increase the accuracy of machining operations in the tilted working plane and of simultaneous machining operations. In part, the control can compensate for temperature-induced deviations through repeated measurements and corrections. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual
OPC UA NC Server (#56-61 / #3-02-1*)	**OPC UA NC Server** These software options include OPC UA, a standardized interface for remote access to the control's data and functions. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection. If more than one parallel connection is required, you need to enable multiple of these software options. If your control features the **SIK2**, you can order this software option multiple times and enable up to ten connections. **Further information:** User's Manual for Setup and Program Run
4 Additional Axes (#77 / #6-01-1*)	**Four additional control loops** **Further information:** "**Control Loop Qty.** (#0-3 / #6-01-1*)", Page 47
Ext. Tool Management (#93 / #2-03-1)	**Extended tool management** This software option extends tool management by the two tables **Tooling list** and **T usage order**. The tables show the following contents: ■ The **Tooling list** shows the tool requirements of the NC program or pallet to be run ■ The **T usage order** shows the tool order of the NC program or pallet to be run **Further information:** User's Manual for Setup and Program Run Extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.
Remote Desk. Manager (#133 / #3-01-1)	**Remote Desktop Manager** This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you don't need to move between different workstations as much, and so increase your efficiency. **Further information:** User's Manual for Setup and Program Run

Software option	Definition and application
Collision Monitoring v2 (#140 / #5-03-2)	**Dynamic Collision Monitoring (DCM)** The machine manufacturer can use this software option to define machine components as collision objects. The control monitors the defined collision objects during all machine movements. The software option includes the following functions: ■ Automatic interruption of program run whenever a collision is imminent ■ Warnings in case of manual axis movements ■ Collision monitoring in Test Run mode ■ Collision monitoring of fixtures ■ Defining a reduced minimum distance between fixture and tool With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or machine downtime. **Further information:** User's Manual for Setup and Program Run
Cross Talk Comp. (#141 / #2-20-1)	**Compensation of axis couplings (CTC)** Using this software option, the machine manufacturer can, for example, compensate for acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adapt. Contr. (#142 / #2-21-1)	**Position Adaptive Control (PAC)** Using this software option, the machine manufacturer can, for example, compensate for position-induced deviations at the tool and thus increase accuracy and dynamic performance.
Load Adapt. Contr. (#143 / #2-22-1)	**Load Adaptive Control (LAC)** Using this software option, the machine manufacturer can, for example, compensate for load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adapt. Contr. (#144 / #2-23-1)	**Motion Adaptive Control (MAC)** Using this software option, the machine manufacturer can, for example, change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Contr. (#145 / #2-30-1)	**Active Chatter Control (ACC)** With this software option the chatter tendency of a machine used for heavy machining can be reduced. The control can use ACC to improve the surface quality of the workpiece, increase the tool life and reduce the machine load. Depending on the type of machine, the metal-removal rate can be increased by more than 25%. **Further information:** User's Manual for Setup and Program Run
Machine Vibr. Contr. (#146 / #2-24-1)	**Vibration damping for machines (MVC)** Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD **Active Vibration Damping** ■ FSC **Frequency Shaping Control**
CAD Model Optimizer (#152 / #1-04-1)	**Optimization of CAD models** This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. **Further information:** User's Manual for Setup and Program Run

Software option	Definition and application
Batch Process Mngr. (#154 / #2-05-1)	**Batch Process Manager (BPM)** This software option makes it easy to plan and execute multiple production jobs. By extending and combining the pallet management and extended tool management functions (#93 / #2-03-1), BPM offers the following additional data, for example: ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs **Further information:** Programming and Testing User's Manual
Component Monitoring (#155 / #5-02-1)	**Component monitoring** This software option enables the automatic monitoring of machine components configured by the machine manufacturer. Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.
Model Aided Setup (#159 / #1-07-1)	**Graphically supported setup** This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions. The control supports you additionally by showing the setup situation and possible touch points in the **Simulation** workspace by means of a 3D model. **Further information:** User's Manual for Setup and Program Run
Opt. Contour Milling (#167 / #1-02-1)	**Optimized contour machining (OCM)** This software option enables trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions. The software option includes the following cycles: ■ Cycle **271 OCM CONTOUR DATA** ■ Cycle **272 OCM ROUGHING** ■ Cycle **273 OCM FINISHING FLOOR** and Cycle **274 OCM FINISHING SIDE** ■ Cycle **277 OCM CHAMFERING** ■ In addition, the control provides **OCM STANDARD FIGURES** for frequently needed contours With OCM you can shorten machining times while at the same time reducing tool wear. **Further information:** "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352

Software option	Definition and application
PTI 8 channels (#3-04-1*)	**Process Tracking Interface PTI** This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with **SIK2**. You can order this software option up to two times, each time selecting up to eight channels for recording. **Further information:** User's Manual for Setup and Program Run

For more information, refer to the integrated help system of RemoTools SDK.
ID 340442-xx

3.3.2 Information on licensing and use

Open-source software

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

To get to the licensing terms on the control:

- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Operating system** tab
- Double-tap or double-click **About HeROS**
- > The control opens the **HEROS License Viewer** window.

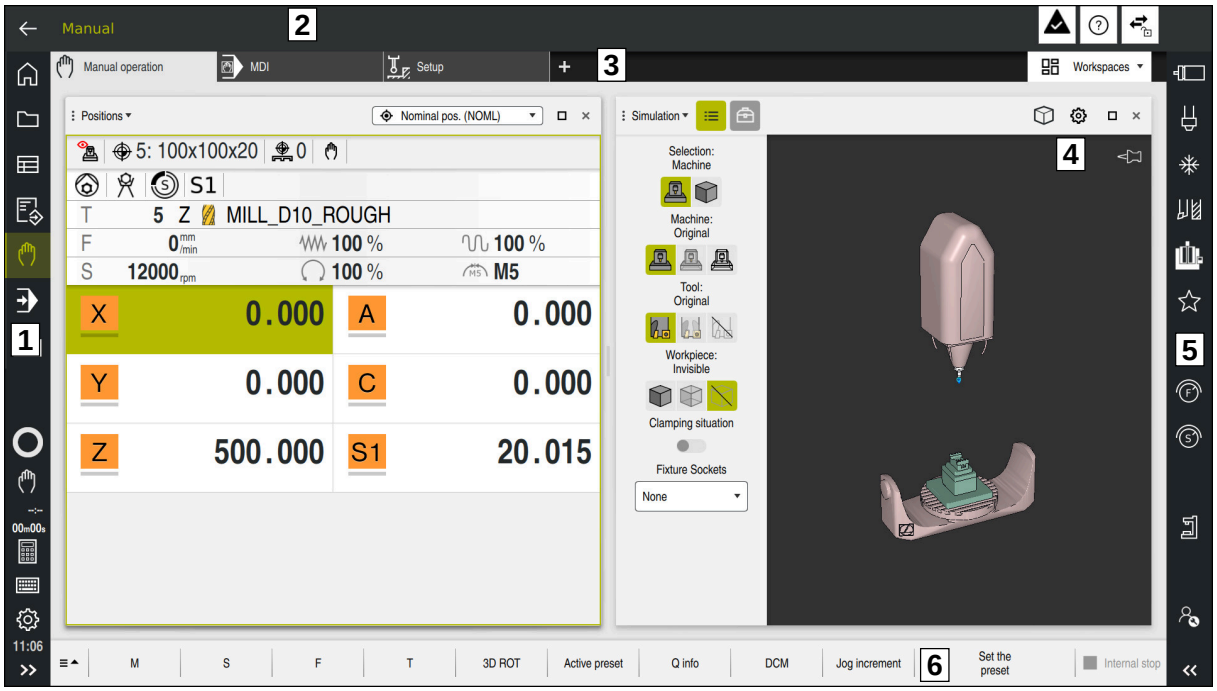
OPC UA

The control software contains binary libraries, to which the terms of use agreed between HEIDENHAIN and Softing Industrial Automation GmbH additionally and preferentially apply.

The control's behavior can be influenced by means of the OPC UA NC Server (#56-61 / #3-02-1*) and HEIDENHAIN DNC (#18 / #3-03-1). Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: User's Manual for Setup and Program Run

3.4 Areas of the control's user interface



User interface of the control in the **Manual operation** application

The control's user interface shows the following areas:

1 TNC bar

- Back

  Use this function to go backwards in the application history since booting the control.
- Operating modes

  Further information: "Overview of the operating modes", Page 55
- Status overview

  Further information: User's Manual for Setup and Program Run
- Calculator

  Further information: Programming and Testing User's Manual
- Screen keyboard
- Settings

  The Settings menu enables you to change the control interface:
  - **Left-hand mode**

    The control swaps the positions of the TNC bar and the machine manufacturer bar.
  - **Dark Mode**

    In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether the **Dark Mode** function is available for selection.

    This manual describes the state of the GUI with inactive **Dark Mode**. Your control might display other colors than shown in this User's Manual.
  - **Font size**

- Date and time

2 Information bar
- Active operating mode
- Message menu
- Symbols

3 Application bar
- Tabs of opened applications

  The maximum number of simultaneously opened applications is limited to ten tabs. If you try to open an eleventh tab, the control shows a message.
- Selection menu for workspaces

  With the selection menu you define which workspaces are open in the active application.

4 Workspaces

5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.

6 Function bar
- Selection menu for buttons

  With the selection menu you define which buttons the control displays in the function bar.
- Button

  With the buttons you activate individual functions of the control.

Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to dim or hide the following user interface elements, for example:
- Operating modes
- Applications
- Workspaces
- Buttons

3.5 Overview of the operating modes

The control provides the following operating modes:

Icon	Operating modes	Further information
	The **Home** operating mode contains the following applications: ■ The **Start/Login** application During the startup process, the control is in the **Start/Login** application. ■ The **Settings** application ■ The **Help** application ■ Applications for machine parameters	 See the User's Manual for Setup and Program Run See the User's Manual for Programming and Testing See the User's Manual for Setup and Program Run
	In the **Files** operating mode the control displays drives, folders and files. You can, for example, create or delete folders or files and can also connect drives.	See the User's Manual for Programming and Testing
	In the **Tables** operating mode you can open various tables and edit them as necessary.	
	In the **Editor** operating mode you can do the following: ■ Create, edit and simulate NC programs ■ Create and edit contours ■ Create and edit pallet tables	See the User's Manual for Programming and Testing
	The **Manual** operating mode contains the following applications: ■ The **Manual operation** application ■ The **MDI** Application ■ The **Setup** application ■ The **Move to ref. point** application ■ The **Retract** application You can move the tool away from the workpiece, for example after a power failure.	 See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run
	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence. You also execute pallet tables in this operating mode.	See the User's Manual for Setup and Program Run
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	See the User's Manual for Setup and Program Run

Icon	Operating modes	Further information
	In the **Machine** operating mode, the machine manufacturers define their own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	

Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to gray out or hide the following user interface elements, for example:

- Operating modes
- Applications
- Buttons

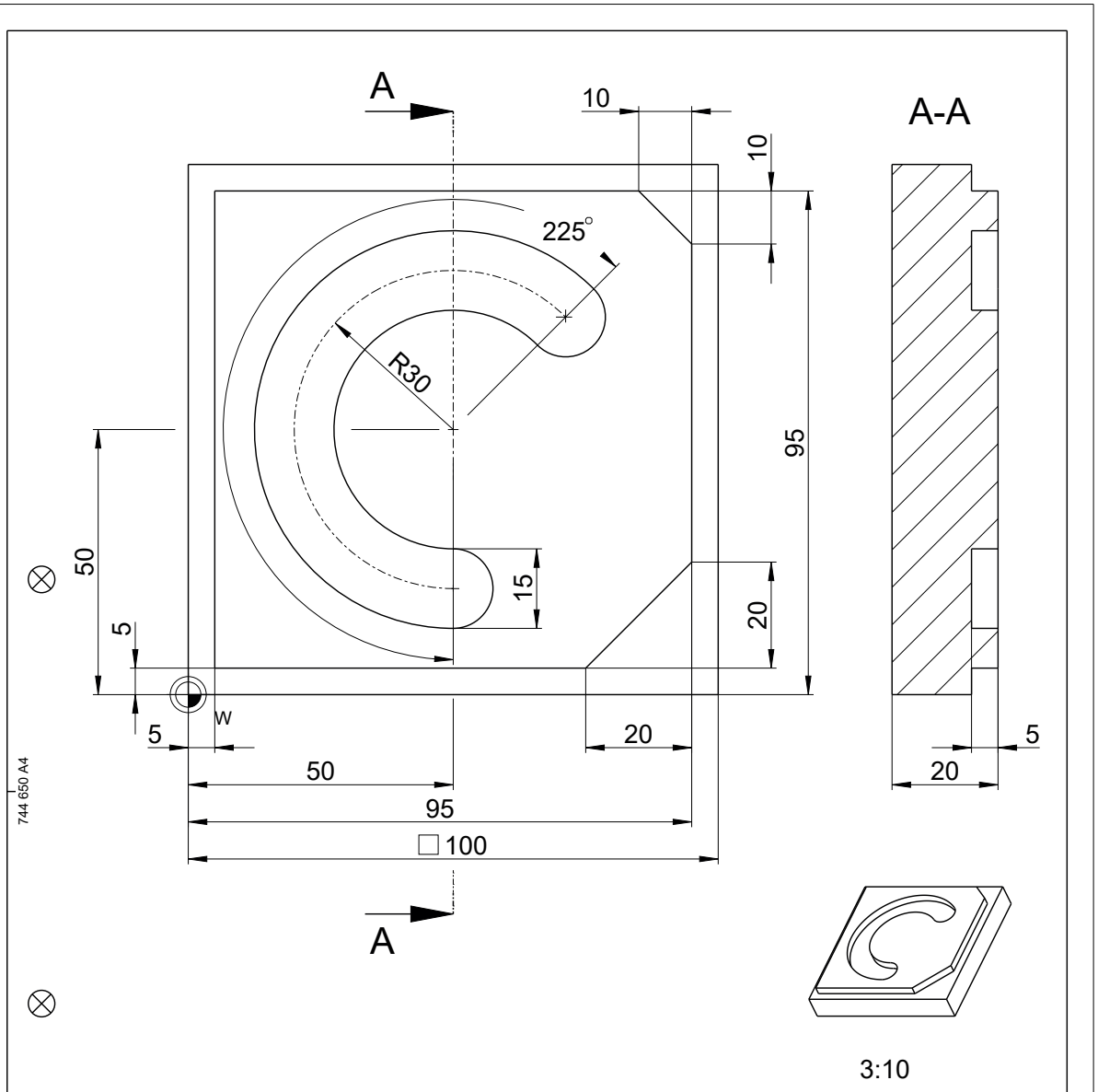
4

First steps

4.1 Programming and simulating a workpiece

4.1.1 Programming the workpiece

Sample task 1338459-00-A-01



			ID number
Text:			Change No. C000941-05 Phase: Nicht-Serie
Original drawing Scale / Format	Platte Plate		Werkstoff: Material:
RoHS / 1:1 / A4			
Maße in mm / Dimensions in mm	Einzelteilzeichnung / Component Drawing		●blanke Flächen/Blank surfaces
Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715 -0.3 +0.3	Allgemeintoleranzen ISO 2768:1989-mH ≦6mm: ±0,2 General tolerances ISO 2768:1989-mH ≦6mm: ±0,2	Tolerierung nach ISO 8015 Tolerances as per ISO 8015	Oberflächen nach ISO 1302:2002 Surfaces as per ISO 1302:2002
		Oberflächenbehandlung: Surface treatment:	

The reproduction, distribution and utilization of this document as well as the communication of its contents to others without express authorization is prohibited. Offenders will be held liable for the payment of damages. All rights reserved in the event of the grant of a patent, utility model or design. (ISO 16016)

HEIDENHAIN	Created	Responsible	Released	Version	Revision	Sheet	Page
DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany	M-TA 17.03.2025			D1358459-00-A-01 Document number			1 of 1

Selecting the Editor operating mode

NC programs are always programmed in the **Editor** operating mode.

Requirement

- It must be possible to select the icon of the operating mode

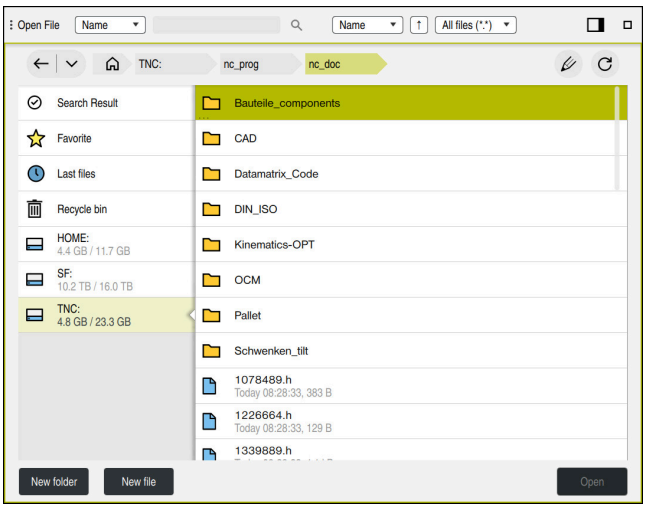
  In order to be able to select the **Editor** operating mode, the control must have already progressed enough during booting that the operating mode icon is no longer dimmed.

Selecting the Editor operating mode

To select the **Editor** operating mode:

- ▶ Select the **Editor** operating mode
- > The control displays the **Editor** operating mode and the most recently opened NC program.

Creating a new NC program



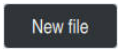
The **Open File** workspace in the **Editor** operating mode

To create an NC program in the **Editor** operating mode:

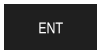
- ▶ Select **Add**
- > The control shows the **Quick selection** and **Open File** workspaces.

- ▶ Select the desired drive in the **Open File** workspace

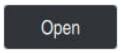
- ▶ Select a folder



- ▶ Select **New file**
- ▶ Enter a file name (e.g., 1338459.h)



- ▶ Confirm with the **ENT** key



- ▶ Select **Open**
- > The control opens a new NC program and the **Insert NC function** window for definition of the workpiece blank.

More detailed information

- The **Open File** workspace
  Further information: User's Manual for Setup and Program Run
- The **Editor** operating mode
  Further information: Programming and Testing User's Manual

Configuring the control's user interface for programming

The **Editor** operating mode gives you several possibilities for writing an NC program.

> The first steps describe the procedure when you are in the **Klartext editor** mode with the **Form** column open.

Opening the Form column

You can open the **Form** column only if an NC program is open.

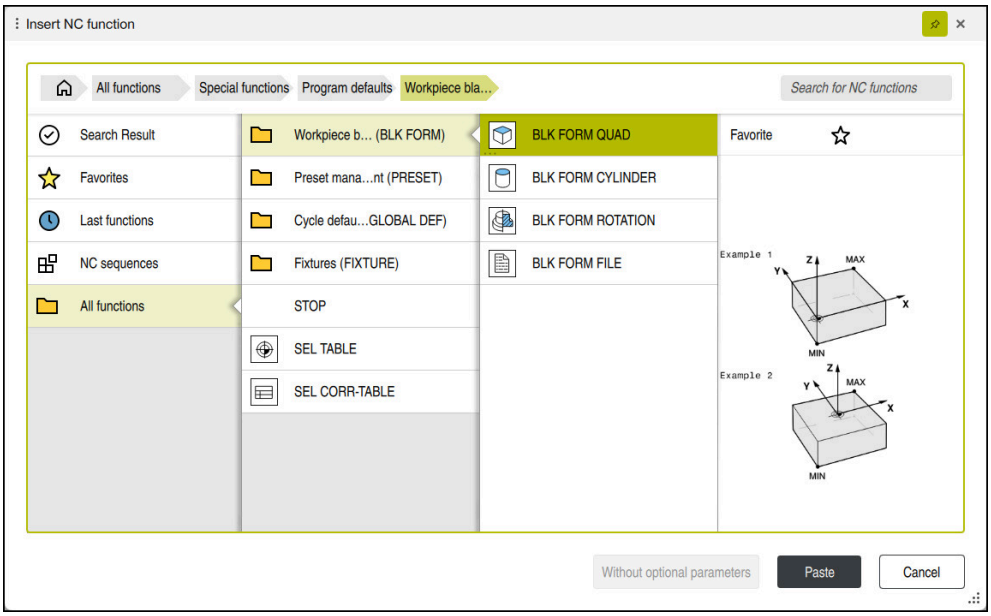
To open the **Form** column:

- ▶ Select **Form**
- > The control opens the **Form** column

Defining the workpiece blank

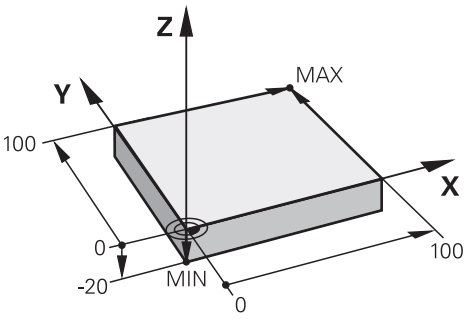
For the NC program you can define a workpiece blank that the control then uses for the simulation. When you create an NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

> If you close the window without selecting a workpiece blank, you can use the **Insert NC function** button to select the workpiece blank definition later.



The **Insert NC function** window for workpiece blank definition

Defining a cuboid workpiece blank



Cuboid workpiece blank with minimum point and maximum point

You define a cuboid through a diagonal in space by entering the minimum point and maximum point relative to the active workpiece preset.

> You can confirm the entries as follows:
> - **ENT** key
> - Right arrow key
> - Click or tap the next syntax element

To define a cuboid workpiece blank:

- Select **BLK FORM QUAD**

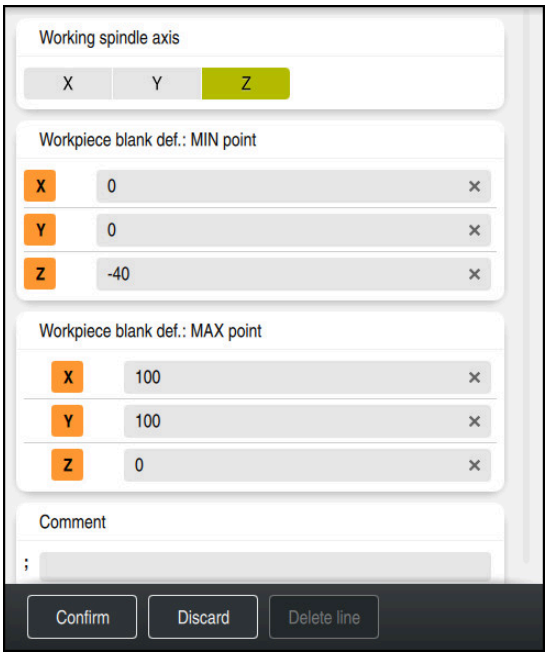
Paste

- Select **Paste**
- > The control inserts the NC block for definition of the workpiece blank.

- Open the **Form** column
- Select the tool axis (e.g., **Z**)
- Confirm your input
- Enter the smallest X coordinate (e.g., **0**)
- Confirm your input
- Enter the smallest Y coordinate (e.g., **0**)
- Confirm your input
- Enter the smallest Z coordinate (e.g., **-20**)
- Confirm your input
- Enter the largest X coordinate (e.g., **100**)
- Confirm your input
- Enter the largest Y coordinate (e.g., **100**)
- Confirm your input
- Enter the largest Z coordinate (e.g., **0**)
- Confirm your input

Confirm

- Select **Confirm**
- > The control concludes the NC block.



The **Form** column with the defined columns

```
0 BEGIN PGM  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 END PGM  MM
```

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

More detailed information

- Inserting the workpiece blank
- Reference points in the machine

Structure of an NC program

Using a uniform structure for an NC program offers the following advantages:

- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program

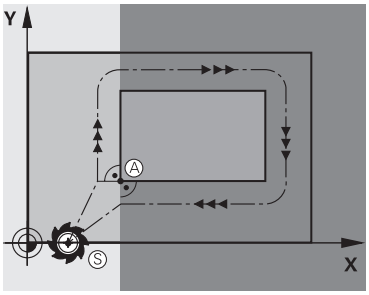
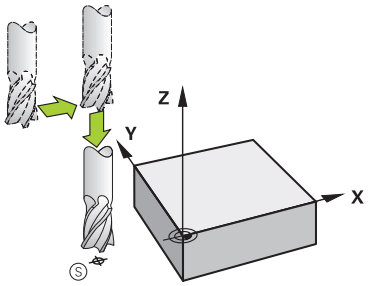
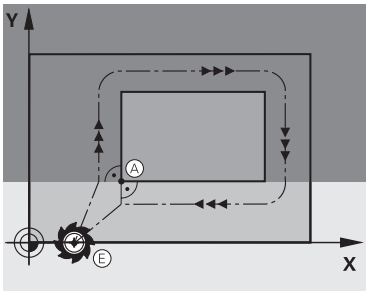
The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks.

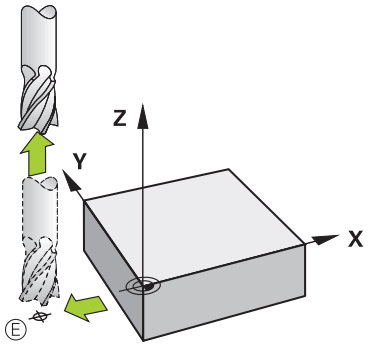
1 **BEGIN PGM** with selection of the unit of measure
2 Define the workpiece blank
3 Call the tool, with the tool axis and the technological data
4 Move the tool to a safe position, and switch the spindle on
5 Pre-position the tool in the working plane, near the first contour point
6 Pre-position the tool in the tool axis, turn coolant on if necessary
7 Approach the contour, activate tool radius compensation if necessary
8 Machine the contour
9 Depart from the contour, turn coolant off
10 Move the tool to a safe position
11 Conclude the NC program
12 **END PGM**

Contour approach and departure

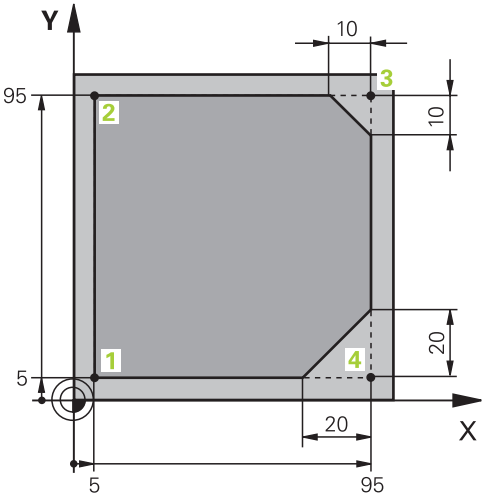
When you program a contour, you need a starting point and end point outside the contour.

The following positions are necessary for contour approach and departure:

Help graphic	Position
	**Starting point** The following preconditions apply for the starting point: ■ No tool radius compensation ■ Approachable without danger of collision ■ Near to the first contour point The graphic shows the following information: If you define the starting point to be in the dark gray area, the contour will be damaged when the first contour point is approached.
	**Approaching the starting point in the tool axis** Before approaching the first contour point, you must position the tool to the working depth in the tool axis. If there is a danger of collision, approach the starting point in the tool axis separately.
	First contour point The control moves the tool from the starting point to the first contour point. You need to program tool radius compensation for the tool movement to the first contour point.
	**End point** The following preconditions apply for the end point: ■ Approachable without danger of collision ■ Near to the last contour point ■ In order to make sure that the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element The graphic shows the following information: If you define the end point to be in the dark gray area, the contour will be damaged when the end point is approached.

Help graphic	Position
	**Departing from the end point in the tool axis** Program the tool axis separately when departing from the end point.
	Identical starting and end points Do not program any tool radius compensation if the starting point and end point are the same. In order to make sure that the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

Programming a simple contour



Workpiece to be programmed

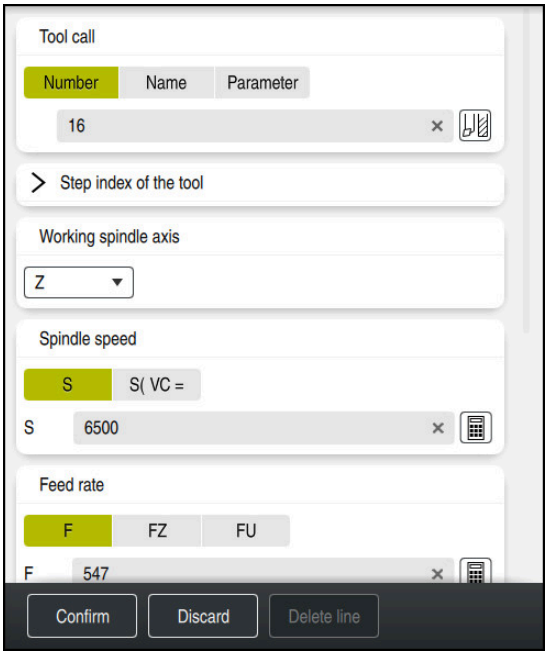
The following texts show you how to mill once at a depth of 5 mm around the contour shown here. You have already defined the workpiece blank.

Further information: "Defining the workpiece blank", Page 61

After you have inserted an NC function, the control shows an explanation about the current syntax element in the dialog bar. You can enter the data directly in the form.

Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.

Calling a tool



The **Form** column with the syntax elements of the tool call

To call a tool:

TOOL CALL

- Select **TOOL CALL**

- Select **Number** in the form
- Enter the tool number (e.g., **16**)
- Select the tool axis **Z**
- Select the spindle speed **S**
- Enter the spindle speed (e.g., **6500**)

Confirm

- Select **Confirm**

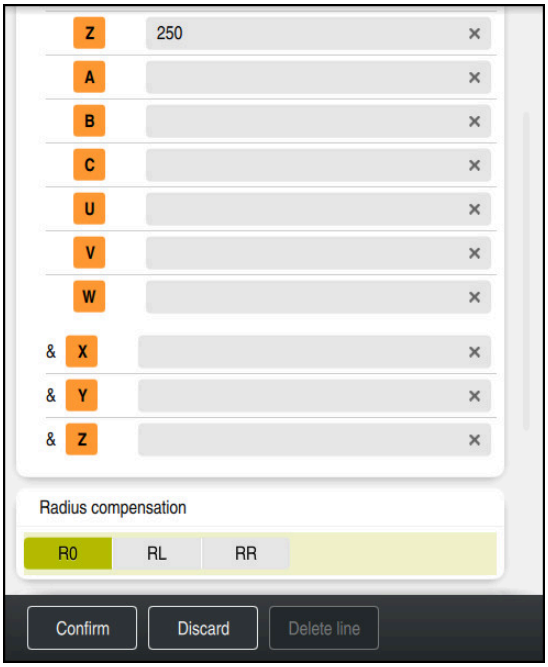
> The control concludes the NC block.

```
3 TOOL CALL 16 Z S6500
```

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

Move the tool to a safe position

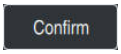


The **Form** column with the syntax elements of a straight line

To move the tool to a safe position:

- ▶ Select the path function **L**

- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- ▶ Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)



- ▶ Select **Confirm**
- > The control concludes the NC block.

```
4 L Z+250 R0 FMAX M3
```

Pre-positioning in the working plane

To pre-position in the working plane:

- Select the path function **L**
- Select **X**
- Enter a value (e.g., **-20**)
- Select **Y**
- Enter a value (e.g., **-20**)
- Select the **FMAX** feed rate
- Select **Confirm**
- The control concludes the NC block.

```
5 L X-20 Y-20 FMAX
```

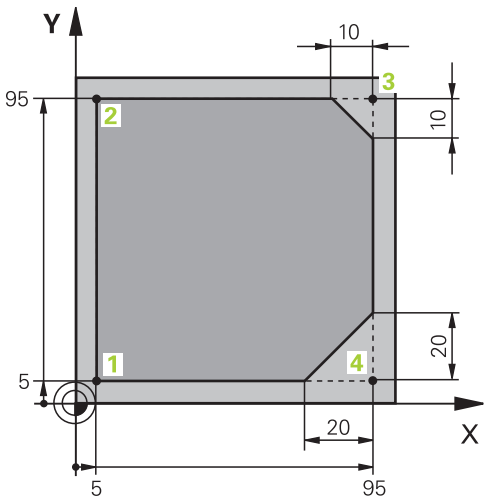
Pre-positioning in the tool axis

To pre-position in the tool axis:

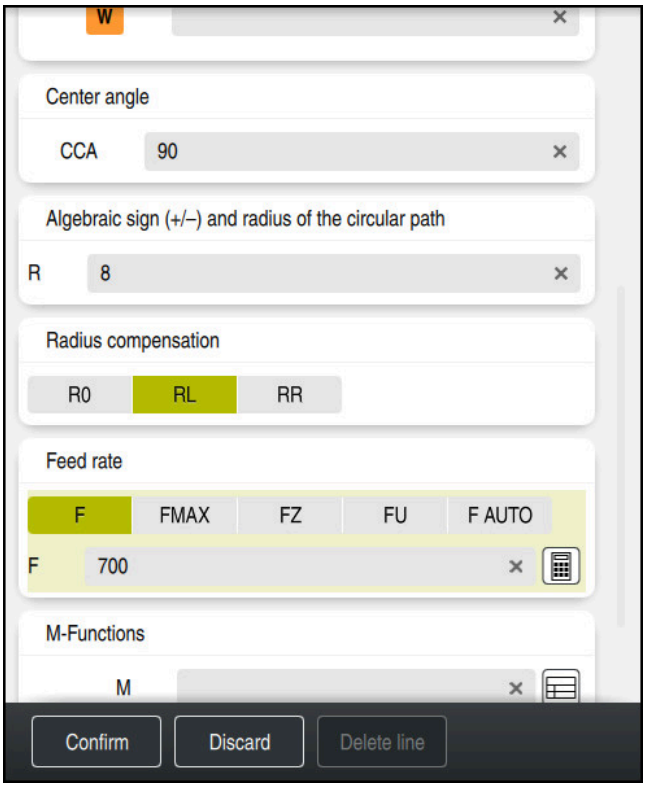
- Select the path function **L**
- Select **Z**
- Enter a value (e.g., **-5**)
- Select the feed rate **F**
- Enter the value for the positioning feed rate (e.g., **3000**)
- If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)
- Select **Confirm**
- The control concludes the NC block.

```
6 L Z-5 F3000 M8
```

Approaching the contour



Workpiece to be programmed



The **Form** column with the syntax elements of an approach function

To approach the contour:

APPR /DEP

- ▶ Select the **APPR DEP** path function
- > The control opens the **Insert NC function** window.

- ▶ Select **APPR**

- ▶ Select an approach function (e.g., **APPR CT**)

Paste

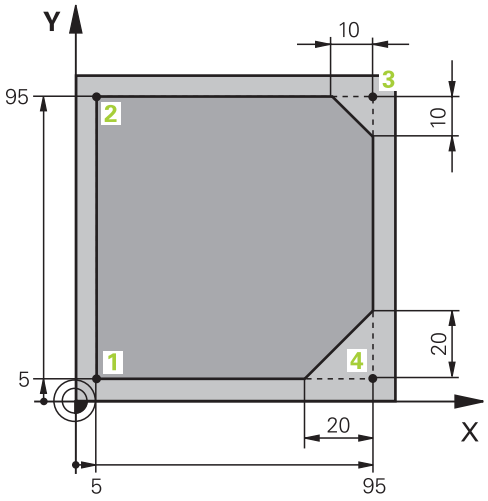
- ▶ Select **Paste**
- ▶ Enter the coordinates of the starting point 1 (e.g., **X 5 Y 5**)
- ▶ For the center angle **CCA**, enter the approach angle (e.g., **90**)
- ▶ Enter the radius of the circular arc (e.g., **8**)
- ▶ Select **RL**
- > The control applies tool radius compensation to the left.
- ▶ Select the feed rate **F**
- ▶ Enter the value for the machining feed rate (e.g., **700**)

Confirm

- ▶ Select **Confirm**
- > The control concludes the NC block.

```
7 APPR CT X+5 Y+5 CCA90 R+8 RL F700
```

Machining a contour



Workpiece to be programmed

To machine the contour:

- Select the path function **L**
- Enter the coordinates of contour point **2** that differ (e.g., **Y 95**)

Confirm

- Conclude the NC block with **Confirm**

> The control applies the changed value and retains all of the other information from the previous NC block.

- Select the path function **L**
- Enter the coordinates of contour point **3** that differ (e.g., **X 95**)

Confirm

- Conclude the NC block with **Confirm**

- Select the path function **CHF**
- Enter the chamfer width (e.g., **10**)

Confirm

- Conclude the NC block with **Confirm**

- Select the path function **L**
- Enter the coordinates of contour point **4** that differ (e.g., **Y 5**)

Confirm

- Conclude the NC block with **Confirm**

- Select the path function **CHF**
- Enter the chamfer width (e.g., **20**)

Confirm

- Conclude the NC block with **Confirm**

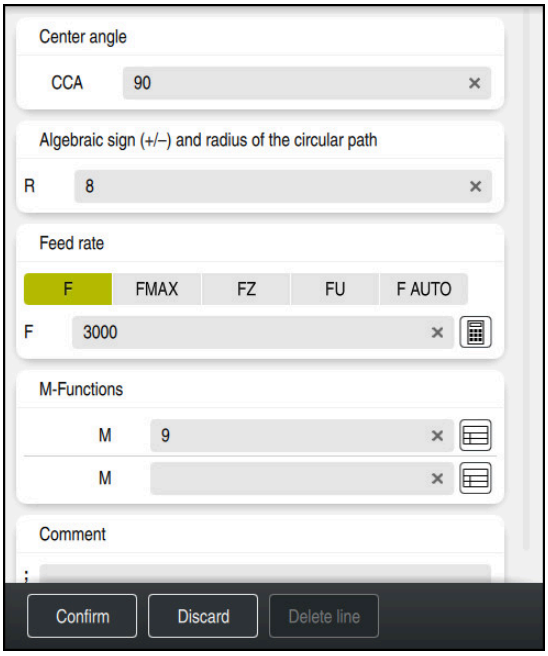
- Select the path function **L**
- Enter the coordinates of contour point **1** that differ (e.g., **X 5**)

Confirm

- Conclude the NC block with **Confirm**

```
8 L Y+95
9 L X+95
10 CHF 10
11 L Y+5
12 CHF 20
13 L X+5
```

Departing from the contour



The **Form** column with the syntax elements of a departure function

To depart from the contour:

APPR /DEP

- Select the **APPR DEP** path function

> The control opens the **Insert NC function** window.

- Select **DEP**

- Select a departure function (e.g., **DEP CT**)

Paste

- Select **Paste**

- For the center angle **CCA**, enter the departure angle (e.g., **90**)
- Enter the departure radius (e.g., **8**)
- Select the feed rate **F**
- Enter the value for the positioning feed rate (e.g., **3000**)
- If needed, enter a miscellaneous function **M**, such as **M9** (turn coolant off)

Confirm

- Select **Confirm**

> The control concludes the NC block.

```
14 DEP CT CCA90 R+8 F3000 M9
```

Programming a machining cycle

The following texts show you how to mill the circular slot of the example task at a depth of 5 mm. You have already defined the workpiece blank and created the outside contour.

Further information: "Sample task 1338459-00-A-01", Page 58

After you have inserted a cycle, you can define the associated values in the cycle parameters. You can program the cycle directly in the **Form** column.

Calling a tool

To call a tool:

TOOL CALL

- Select **TOOL CALL**

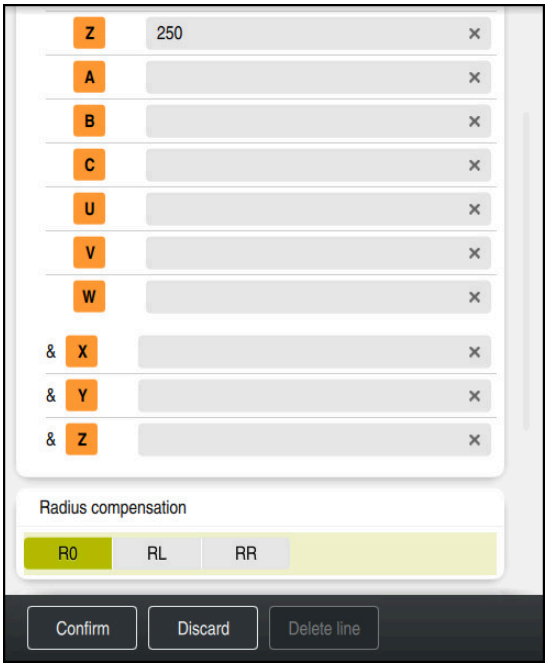
- Select **Number** in the form
- Enter the tool number (e.g., **6**)
- Select the tool axis **Z**
- Select the spindle speed **S**
- Enter the spindle speed (e.g., **6500**)

Confirm

- Select **Confirm**
- The control concludes the NC block.

```
16  TOOL CALL 6 Z S6500
```

Moving the tool to a safe position

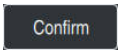


The **Form** column with the syntax elements of a straight line

To move the tool to a safe position:

- ▶ Select the path function **L**

- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- ▶ Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)



- ▶ Select **Confirm**
- > The control concludes the NC block.

```
17 L  Z+250 R0 FMAX M3
```

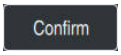
Pre-positioning in the working plane

To pre-position in the working plane:

- Select the path function **L**

- Select **X**
- Enter a value (e.g., **+50**)

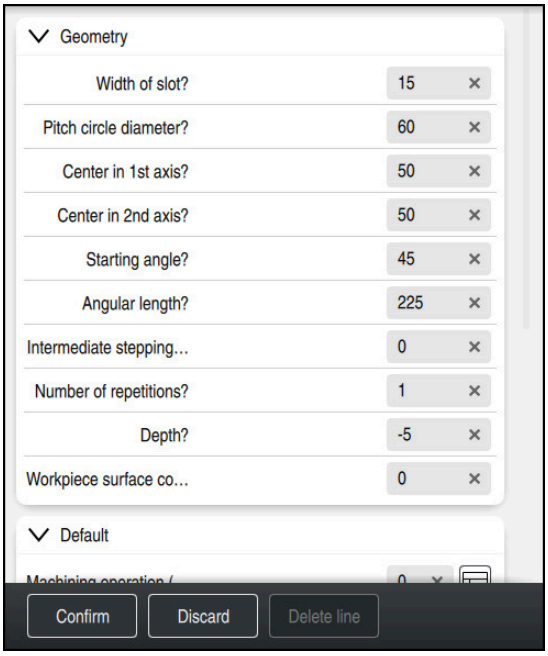
- Select **Y**
- Enter a value (e.g., **+50**)
- Select the **FMAX** feed rate



- Select **Confirm**
- > The control concludes the NC block.

```
18 L  X+50 Y+50 FMAX
```

Defining a cycle



The **Form** column with possibilities for entering cycle information

To define the circular slot:

CYCL DEF

- ▶ Select the **CYCL DEF** key

> The control opens the **Insert NC function** window.

- ▶ Select Cycle **254 CIRCULAR SLOT**

Paste

- ▶ Select **Paste**

> The control inserts the cycle.

- ▶ Open the **Form** column
- ▶ Enter all input values in the form

Confirm

- ▶ Select **Confirm**

> The control saves the cycle.

```
19 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0              ;MACHINING OPERATION ~
    Q219=+15             ;SLOT WIDTH ~
    Q368=+0.1            ;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=+45             ;STARTING ANGLE ~
    Q248=+225            ;ANGULAR LENGTH ~
    Q378=+0              ;STEPPING ANGLE ~
    Q377=+1              ;NR OF REPETITIONS ~
    Q207=+500            ;FEED RATE MILLING ~
    Q351=+1              ;CLIMB OR UP-CUT ~
    Q201=-5              ;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
```

Calling a cycle

To call the cycle:

CYCL CALL

- Select **CYCL CALL**

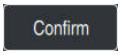
```
20 CYCL CALL
```

Moving the tool to a safe position and concluding the NC program

To move the tool to a safe position:

- Select the path function **L**

- Select **Z**
- Enter a value (e.g., **250**)
- Select tool radius compensation **R0**
- Select the **FMAX** feed rate
- Enter miscellaneous function **M** (e.g., **M30**, end of program run)



- Select **Confirm**
- > The control concludes the NC block and the NC program.

```
21  L  Z+250 R0 FMAX M30
```

More detailed information

- Working with cycles
  Further information: "Working with cycles", Page 84

Solution 1338459-00-A-01

If you follow all steps described here, you will obtain the following NC program:

```
0 BEGIN PGM "1338459-00-A-01" 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 16 Z S6500
4 L Z+250 R0 FMAX M3
5 L X-20 Y-20 FMAX
6 L Z-5 F3000 M8
7 APPR CT X+5 Y+5 CCA90 R+8 RL F700
8 L Y+95
9 L X+95
10 CHF 10
11 L Y+5
12 CHF 20
13 L X+5
14 DEP CT CCA90 R+8 F3000 M9
15 L Z+250 R0 FMAX
16 TOOL CALL 6 Z S6500
17 L Z+250 R0 FMAX M3
18 L X+50 Y+50 FMAX
19 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0          ;ROUGHING AND FINISHING ~
    Q219=+15         ;SLOT WIDTH ~
    Q368=+0.1        ;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=+45        ;STARTING ANGLE ~
Q248=+225       ;ANGULAR LENGTH ~
Q378=+0         ;STEPPING ANGLE ~
Q377=+1         ;NR OF REPETITIONS ~
Q207=+500       ;FEED RATE MILLING ~
Q351=+1         ;CLIMB MILLING ~
Q201=-5         ;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         ;RECIPROCATING PLUNGE ~
Q385=+500       ;FINISHING FEED RATE ~
Q439=+0         ;CENTER POINT PATH
20 CYCL CALL
21 L Z+250 R0 FMAX
22 M30
23 END PGM "1338459-00-A-01" MM
```

4.1.2 Simulating a workpiece

Configuring the control's user interface for simulation

In the **Editor** operating mode you can test NC programs graphically. The control simulates the active NC program in the **Program** workspace.

In order to simulate the NC program you must open the **Simulation** workspace.

> For the simulation you can close the **Form** column to get a better view of the NC program and the **Simulation** workspace.

Opening the Simulation workspace

You can open additional workspaces in the **Editor** operating mode only if an NC program is open.

To open the **Simulation** workspace:

- In the application bar, select **Workspaces**
- Select **Simulation**

> The control then additionally displays the **Simulation** workspace.

> You can also open the **Simulation** workpiece with the **Test Run** operating mode key.

Configuring the Simulation workspace

You can simulate the NC program without needing to enter any special settings. However, an adjustment to the simulation speed is recommended for best viewing of the simulation.

To adjust the speed of the simulation:

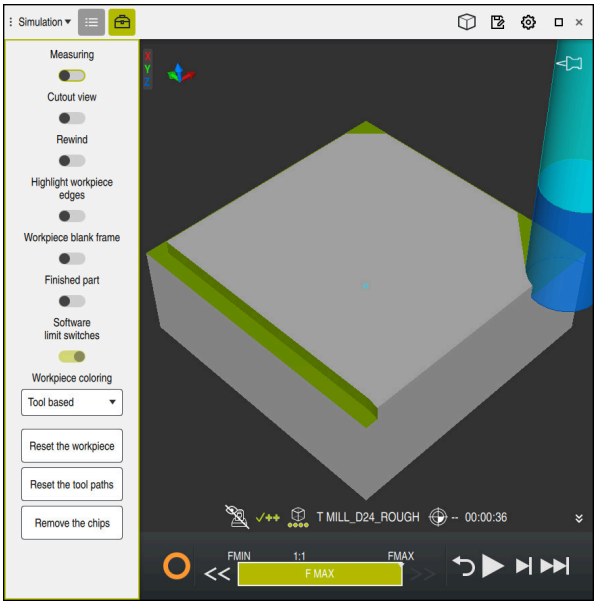
- ▶ Use the slider to select the factor (e.g., **5.0 * T**)
- > The control then performs the subsequent simulation at five times the speed of the programmed feed rate.

If you use different tables, such as tool tables, for program run and the simulation, then you can define the tables in the **Simulation** workspace.

Simulating an NC program

You can test the NC program in the **Simulation** workspace.

Starting the simulation



The **Simulation** workspace in the **Editor** operating mode

To start the simulation:

▶

- ▶ Select **Start**
- > The control might ask whether the file should be saved.

Save

- ▶ Select **Save**
- > The control starts the simulation.
- > The control uses the **Control-in-operation** symbol to show the simulation status.

Definition

Control-in-operation:
The control uses the **Control-in-operation** icon to show the current simulation status in the action bar and on the tab of the NC program:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

5

NC and programming fundamentals

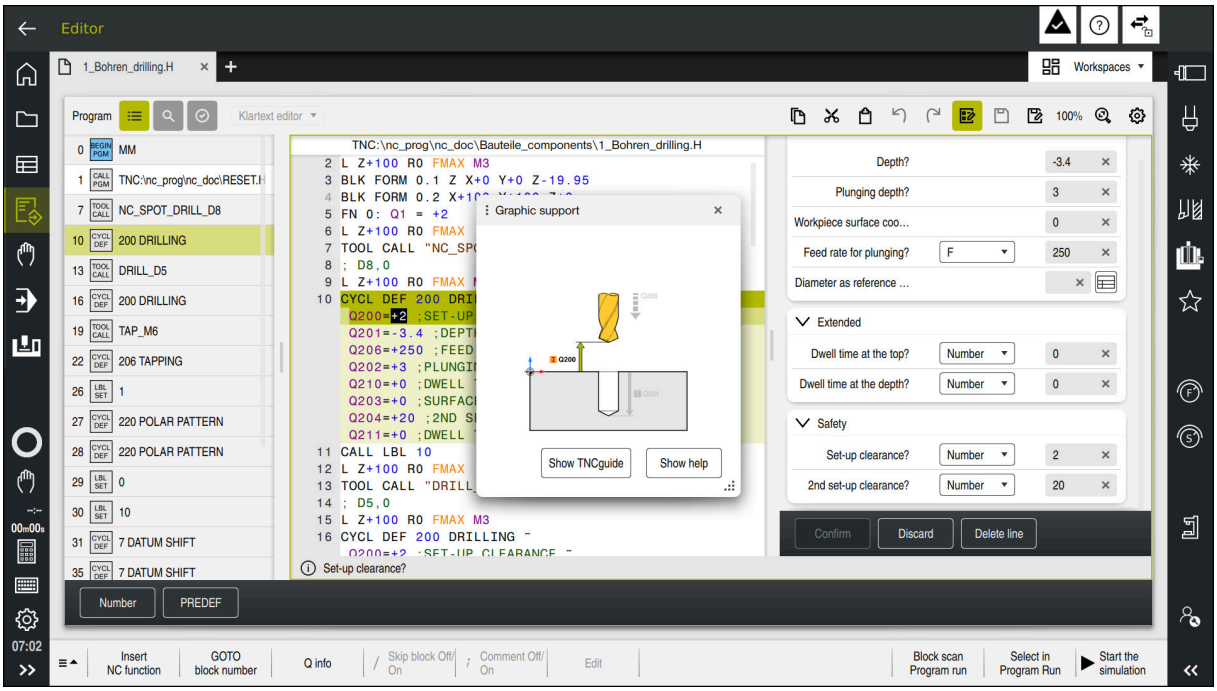
5.1 Working with cycles

5.1.1 General information on cycles

General

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.



Cycles are stored on the control as subprograms. The cycles can be used to execute different machining operations, thereby enabling the easy and efficient programming of recurring machining operations. Most cycles use Q parameters as transfer parameters. The control offers cycles for the following machining technologies:

- Drilling processes
- Thread machining
- Milling operations such as pockets and studs or even contours
- Cycles for coordinate transformation

NOTICE

Danger of collision!

Cycles sometimes perform complex machining movements. There is therefore a risk of collision!

- ▶ Simulate your program before executing it

NOTICE

Danger of collision!

You can program variables as input values in HEIDENHAIN cycles. If you define cycle parameters with variables, then use only the input range of the cycle parameter. Otherwise, collisions may occur.

- Use only the input range of the cycle parameter
- Pay attention to the HEIDENHAIN documentation
- Check the machining sequence using a simulation

In inch programs, the feed rate for cycles must be defined in 0.1 inch/min.

Optional parameters

HEIDENHAIN continuously further develops the extensive package of cycles. Additional cycle parameters for cycles may become available with every new software version. These new Q parameters are **optional parameters**, some of which were not contained in previous software versions. These parameters are located at the end of the cycle definition.

To see which optional Q parameters were added to your current software version, refer to the "New and modified functions" overview.

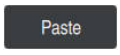
You have the following alternatives when inserting a cycle:

- Insert the cycle with all optional parameters. Every optional parameter can be hidden using the **NO ENT** key.
- Insert the cycle with only its required parameters. In this case, you insert the cycle without optional parameters.

If you hide a Q parameter or insert a cycle with only its required parameters, then the values of the optional Q parameters that are not shown are their default values.

To remove optional parameters in the NC program:

- Select **Insert NC function**
- The control opens the **Insert NC function** window.
- Select the desired cycle



- Select **Paste**
- The control inserts the cycle definition with all cycle parameters into the NC program.

- Navigate to the optional cycle parameters by pressing the arrow key that points to the right

- Press the **NO ENT** key to hide the optional cycle parameter

Insert the cycle with only its required parameters:

- Select **Insert NC function**
- The control opens the **Insert NC function** window.
- Select the desired cycle

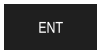
- Select **Without optional parameters**
- The control inserts the cycle definition without optional cycle parameters into the NC program.

If you unintentionally hide an optional parameter or need it again at a later time, then you can once again show and edit it within the cycle.

To show optional parameters:

- Edit cycle

- Navigate to the desired optional cycle parameters by pressing the arrow key that points to the right
- Change the default value as needed



- Confirm the optional cycle parameter by pressing **ENT**

- Save the change by pressing **END**

Compatibility

The TNC7 basic can usually execute the NC programs of predecessor controls starting with the TNC 150 B and later even though the cycles have been extended to include additional optional parameters.

This is possible thanks to the stored default value. To execute a current NC program on an older control or software version, remove the optional cycle parameters. This gives you a backward-compatible NC program. If NC blocks contain invalid elements, then the control indicates that these elements are ERROR blocks.

Defining cycles

Cycles can be defined in several ways.

Inserting via NC function:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Inserting machining cycles via the CYCL DEF key:

CYCL DEF

- ▶ Press the **CYCL DEF** key
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Inserting touch-probe cycles via the TOUCH PROBE key:

TOUCH PROBE

- ▶ Press the **TOUCH PROBE** soft key
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Navigation in the cycle

Key	Function
▸	Navigation within the cycle: Jump to next parameter
◂	Navigation within the cycle: Jump to previous parameter
▾	Jump to the same parameter in the next cycle
▴	Jump to the same parameter in the previous cycle

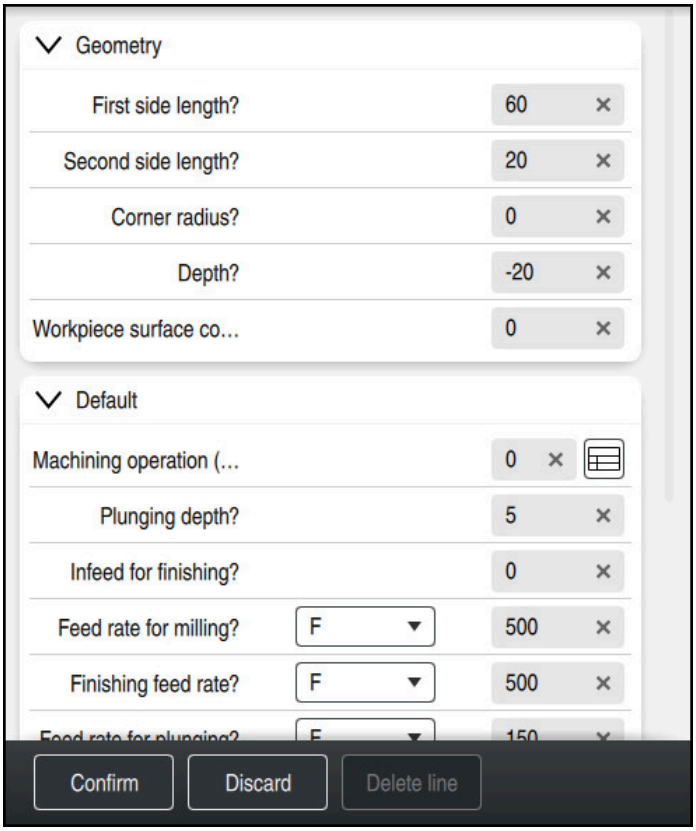
For some cycle parameters, the control provides selectable choices via the action bar or the form.

If an input option specifying a defined behavior is stored in certain cycle parameters, you can open a selection list with the **GOTO** key or in the form view. For example, in Cycle **200 DRILLING** the **Q395 DEPTH REFERENCE** parameter provides the following options:

- 0 | Tool tip
- 1 | Cutting edge corner

Cycle input form

The control provides a **FORM** for various functions and cycles. This **FORM** allows you to enter various syntax elements or cycle parameters.



The control allocates the cycle parameters in the **FORM** to groups based on their functions (e.g., geometry, standard, advanced, safety). The control provides selection possibilities for different cycle parameters via switches, for example. The control displays the currently edited cycle parameter in color.

After you have defined all required cycle parameters, you can confirm your input and conclude the cycle.

To open the form:

- Select the **Editor** operating mode
- Select the desired **Program**
- Select **FORM** via the title bar

If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

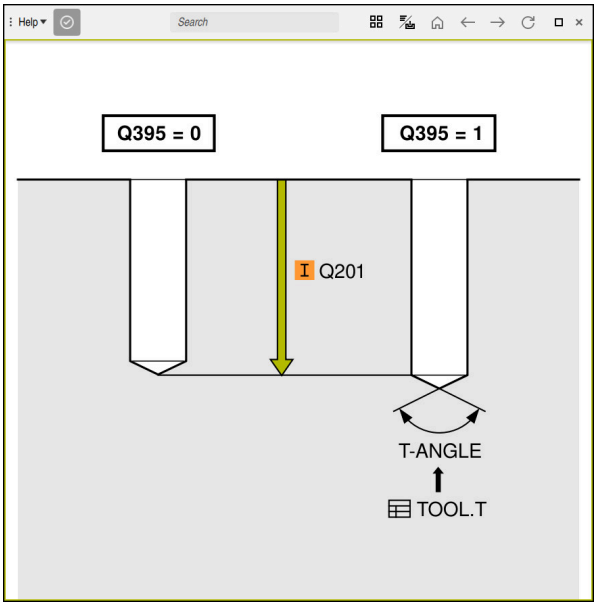
Help graphics

When you are editing a cycle, the control shows a help graphic for the current Q parameters. The size of the help graphic depends on the size of the **Program** workspace.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click on the help graphic, the control maximizes the help graphic.

If the **Help** workspace is active, the control will display the help graphic in this area instead of showing it in the **Program** workspace.



The **Help** workspace with a help graphic for a cycle parameter

Calling cycles

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the machining cycle that was defined last in the NC program.

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (only required for simulation)
- Tool call
- Spindle direction of rotation (miscellaneous function **M3**/**M4**)
- Cycle definition (**CYCL DEF**)

> For some cycles, additional requirements must be observed. They are detailed in the descriptions and overview tables for each cycle.

You can program the cycle call in the following ways:

Syntax	Further information
CYCL CALL	Page 90
CYCL CALL PAT	Page 90
CYCL CALL POS	Page 91
M89/M99	Page 91

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the most recently defined machining cycle once. The starting point of the cycle is the position that was programmed most recently before the **CYCL CALL** block.

Insert NC function

- Select **Insert NC function**

or

CYCL CALL

- Press the **CYCL CALL** key

> The control opens the **Insert NC function** window.

- Select **CYCL CALL M**
- Define **CYCL CALL M** and add an M function, if necessary

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 123

Further information: "Point tables", Page 120

Insert NC function

- Select **Insert NC function**

or

CYCL CALL

- Press the **CYCL CALL** key

> The control opens the **Insert NC function** window.

- Select **CYCL CALL PAT**
- Define **CYCL CALL PAT** and add an M function , if necessary

Calling a cycle with CYCL CALL POS

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

Insert NC function

- Select **Insert NC function**

or

CYCL CALL

- Press the **CYCL CALL** key

> The control opens the **Insert NC function** window.

- Select **CYCL CALL POS**
- Define **CYCL CALL POS** and add an M function as needed

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 notes

- 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 cycles with additional functions

M99

The **M99** miscellaneous function calls the most recently defined machining cycle once. **M99** is effective blockwise and at the end of the block (e.g., after the traverse movement)

Example

```
11 CYCL DEF 257 CIRCULAR STUD
...
12 L X+50 Y+50 R0 FMAX M99
```

The control traverses at **FMAX** to the position **X+50** and **Y+50**. Then the control calls Machining Cycle**257 CIRCULAR STUD** with **M99**.

M89

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

You can cancel **M89** with the following functions:

- **M99** at the last position
- New machining cycle with **CYCL DEF**

Defining and calling an NC program as cycle

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

To define an NC program as a cycle:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.

CYC

- ▶ Select **SEL CYCLE**
- ▶ Select file name, string parameter, or file

To call an NC program as a cycle:

CYCL CALL

- ▶ Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.

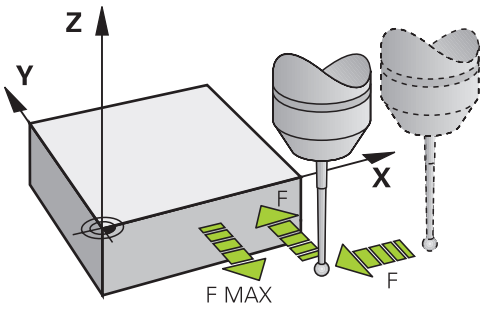
  or
- ▶ Program **M99**

- 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.
- **CYCL CALL PAT** and **CYCL CALL POS** use a positioning logic before the respective cycle is executed. With respect to the positioning logic, the **SEL CYCLE** and Cycle **12 PGM CALL** show the same behavior. In point pattern cycles, the clearance height for approaching is calculated based on:
  - The maximum Z position at the start of the pattern
  - 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.

5.1.2 General information about touch probe cycles

Description of function

- Refer to your machine manual.
- The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.
- The control's full range of functions is available only if the **Z** tool axis is used.
- Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.



The touch-probe functions allow you to determine workpiece misalignment and compensate for it as well as set presets on the workpiece and measure the workpiece.

Whenever the control executes a touch probe cycle, the 3D touch probe approaches the workpiece parallel to a machine axis. This is also true during an active basic rotation or with a tilted working plane.

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

When the probe stylus contacts the workpiece,

- the 3D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the control displays an error message (distance: **DIST** from touch probe table).

5.1.3 Machine-specific cycles

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:

Cycle-number range	Description
300 to **399**	Machine-specific cycles that are to be selected through the **CYCL DEF** key
500 to **599**	Machine-specific touch probe cycles that are to be selected through the **TOUCH PROBE** key

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections, resulting in undesired behavior. Danger of collision during machining!

- Only use variable ranges recommended by HEIDENHAIN
- Do not use pre-assigned variables
- Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- Check the machining sequence using the simulation

Further information: "Calling cycles", Page 90

Further information: Programming and Testing User's Manual

5.1.4 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 179
■ Boring	Page 214
■ Counterboring, centering	
■ Tapping	Page 221
■ Thread milling	Page 235
Pockets/studs/slots	
■ Pocket milling	Page 264
■ Stud milling	Page 290
■ Slot milling	
■ Face milling	Page 390
Coordinate transformations	
■ Mirroring	Page 416
■ Rotating	
■ Magnifying / Reducing	
SL cycles	
■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours	Page 310
■ Cylinder surface machining	Page 446
■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours	Page 352
Point patterns	
■ Bolt hole circle	Page 134
■ Linear hole pattern	
■ Data Matrix code	
Special cycles	
■ Dwell time	Page 430
■ Oriented spindle stop	
■ Tolerance	
■ Ascertain the load	
■ Program call	Page 98
■ Engraving	Page 407

Measuring cycles

Cycle group	Further information
Rotation ■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual
Preset/Position ■ Rectangle, inside or outside ■ Circle, inside or outside ■ Corner, inside or outside ■ Center of bolt circle, slot or ridge ■ Touch probe axis or single axis ■ Four holes	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual
Measuring ■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual
Special cycles ■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing ■ Extrusion probing	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual
Calibrating the touch probe ■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual
Measuring kinematics ■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual
Measuring the tool (TT) ■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT	**Further information:** Measuring Cycles for Workpieces and Tools User's Manual

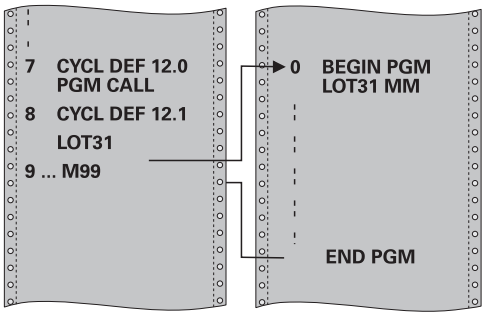
6

Programming techniques

6.1 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.

Related topics

- Calling external NC programs
  Further information: Programming and Testing User's Manual

Note

- You can execute this cycle in the following operating modes: **FUNCTION MODE MILL**.
- 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.

6.1.1 Cycle parameters

Help graphic	Parameter
	Program name Enter the name of the NC program to be called and, if necessary, the path where it is located, Use the Select File Select in the action bar of the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare NC program 1_Plate.h as a cycle and call it with M99

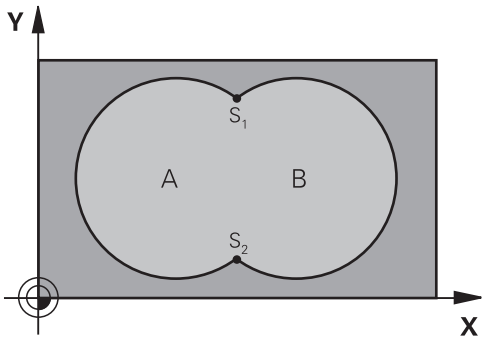
```
11 CYCL DEF 12.0 PGM CALL
12 CYCL DEF 12.1 PGM TNC:\nc_prog\demo\OCM\1_Plate.h
13 L  X+20  Y+50 R0 FMAX M99
```

7

Contour and point definitions

7.1 Superimposing contours

7.1.1 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.

Related topics

- Cycle 14 **CONTOUR**
  Further information: "Cycle 14 CONTOUR ", Page 106
- SL cycles
  Further information: "Milling contours with SL cycles ", Page 310
- OCM cycles
  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352

7.1.2 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.

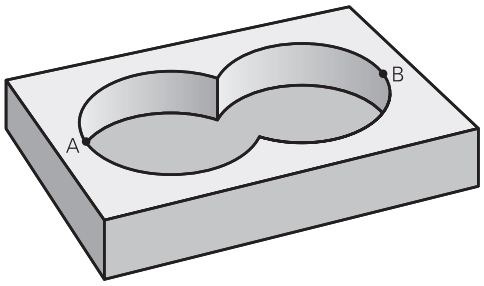
Subprogram 1: Pocket A

```
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
```

7.1.3 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

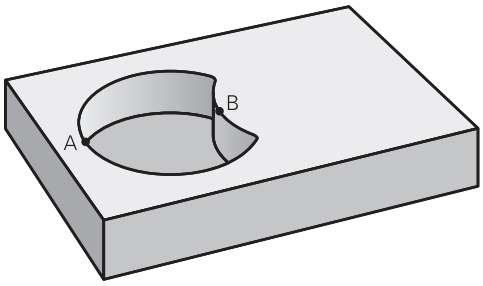
Surface A

```
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
```

7.1.4 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.

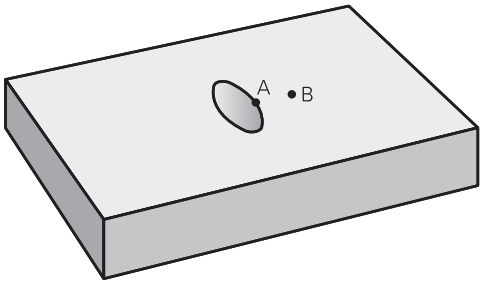
Surface A

```
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 RR
18 CC  X+65  Y+50
19 C  X+40  Y+50 DR-
20 LBL 0
```

7.1.5 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
```

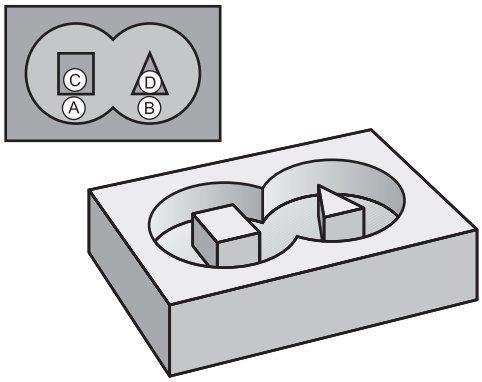
Surface B

```
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
```

7.2 Cycle 14 CONTOUR

ISO programming
G37

Application



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

Related topics

- Simple contour formula
  Further information: "Simple contour formula", Page 107
- Complex contour formula
  Further information: "Complex contour formula", Page 111
- Superimposing contours
  Further information: "Superimposing contours", Page 102

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **14** is DEF-active which means that it takes effect as soon as it is defined in the NC program.
- You can list up to 12 subprograms (subcontours) in Cycle **14**.

7.2.1 Cycle parameters

Help graphic	Parameter
	Label numbers for contour? 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 **END** key. Up to 12 subprogram numbers are possible. Input: **0...65535**

Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL1 /2
```

7.3 Simple contour formula

7.3.1 Fundamentals

Using simple contour formulas, you can easily combine up to nine subcontours (pockets or islands) to program a particular contour. The control calculates the complete contour from the selected subcontours.

Related topics

- Superimposing contours
  Further information: "Superimposing contours", Page 102
- Complex contour formula
  Further information: "Complex contour formula", Page 111
- Cycle 14 **CONTOUR**
  Further information: "Cycle 14 CONTOUR ", Page 106
- SL cycles
  Further information: "Milling contours with SL cycles ", Page 310
- OCM cycles
  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352

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
```

The memory capacity for programming an SL cycle (all contour description programs) is limited to **100 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 areas 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**.

7.3.2 Entering a simple contour formula

You can use the selection possibility in the action bar or in the form to interlink various contours in a mathematical formula.

Proceed as follows:

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **CONTOUR DEF**
- > The control opens the dialog for entering the contour formula.
- ▶ Enter the first subcontour **P1**
- ▶ Select the **P2** pocket or **I2** island selection possibility
- ▶ Enter second subcontour
- ▶ If needed, enter the depth of the second subcontour.
- > Carry on with the dialog as described above until you have entered all subcontours.
- ▶ Define void areas **V** as needed

> 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:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a QS parameter
LBL	■ Number ■ Name ■ Parameter	Define the number, name or variable of a label

Example:

```
11 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 DEPTH5 V1 = LBL 3
```

Programming notes:

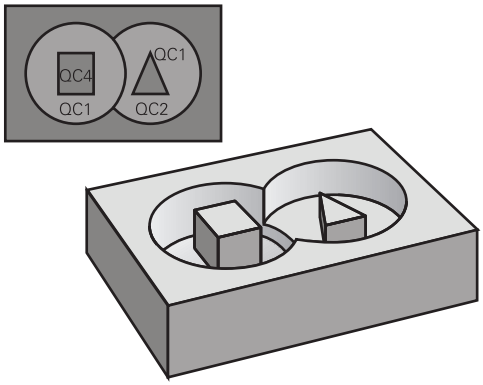
- The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth Therefore, always start programming the subcontour with the deepest pocket.
- If the contour is defined as an island, the control interprets 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 in effect 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.

7.3.3 Machining contours with SL or OCM cycles

The entire contour is machined with the SL cycles (see "Milling contours with SL cycles ", Page 310) or the OCM cycles (see "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352).

7.4 Complex contour formula

7.4.1 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 or subprograms. 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.

Related topics

- Superimposing contours
  Further information: "Superimposing contours", Page 102
- Simple contour formula
  Further information: "Simple contour formula", Page 107
- Cycle 14 **CONTOUR**
  Further information: "Cycle 14 CONTOUR ", Page 106
- SL cycles
  Further information: "Milling contours with SL cycles ", Page 310
- OCM cycles
  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352

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 **100 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
...
```

7.4.2 Selecting an NC program with contour definition

With the **SEL CONTOUR** function, you select an NC program with contour definitions, from which the control extracts the contour descriptions:

Proceed as follows:

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.

- ▶ Select **SEL CONTOUR**
- > The control opens the dialog for entering the contour formula.
- ▶ Definition of the contour

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a QS parameter
LBL	■ Number ■ Name ■ Parameter	Define the number, name or variable of a label

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.
- Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR** is no longer necessary if you use **SEL CONTOUR**.

7.4.3 Defining a contour description

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:

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **DECLARE CONTOUR**
- > The control opens the dialog for entering the contour formula.
- ▶ Enter the number for the contour designator **QC**
- ▶ Defining a contour description

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select "File selection"
QS		Define the number of a QS parameter

Programming notes:

- With the entered contour designators **QC** you can include the various contours in the contour formula.
- 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.
- 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.

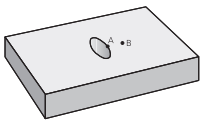
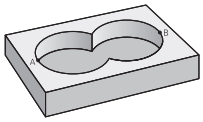
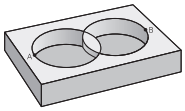
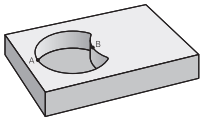
  Further information: "Simple contour formula", Page 107

7.4.4 Entering a complex contour formula

You can use the contour formula function to interlink various contours in a mathematical formula.

Insert NC function

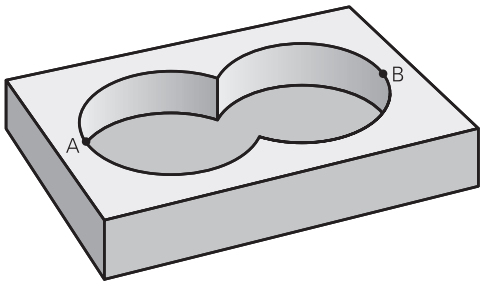
- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **Contour formula QC**
- > The control opens the dialog for entering the contour formula.
- ▶ Enter the number for the contour designator **QC**
- ▶ Entering a contour formula

Help graphic	Input	Mathematical function	Example
	**&**	Intersected with	**QC10 = QC1 & QC2**
	**|**	Joined with	**QC10 = QC1 | QC2**
	**^**	Joined with, but w/o intersection	**QC10 = QC1 ^ QC2**
	****	Without	**QC10 = QC1 \ QC2**
	(	Opening parenthesis	**QC10 = QC1 & (QC2 | QC3)**
	)	Closing parenthesis	**QC10 = QC1 & (QC2 | QC3)**
		Define a single contour	**QC10 = QC1**

The control provides the following options to enter formulas:

- Auto-complete
  Further information: Programming and Testing User's Manual
- Pop-up keyboard for formula input from the action bar or from within the form
- Formula input mode of the virtual keyboard
  Further information: Programming and Testing User's Manual

7.4.5 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.

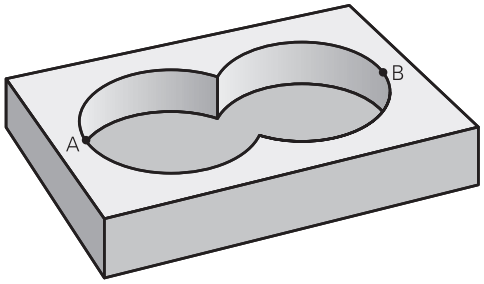
Contour description program 1: Pocket A

```
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



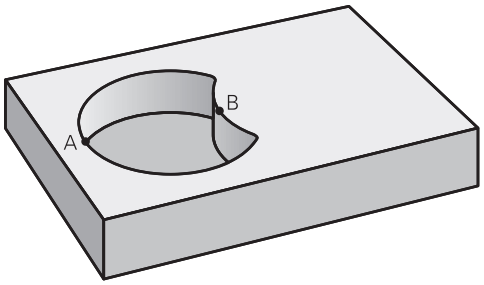
Both areas A and B are to be machined, including the overlapping area:

- 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



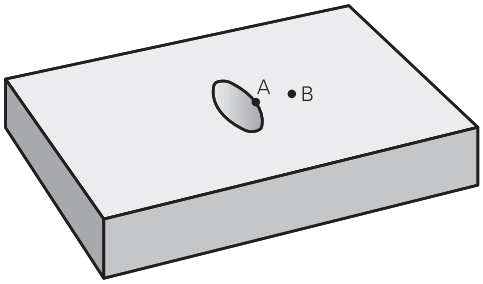
Area A is to be machined without the portion overlapped by B:

- 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 using 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
* - ...
```

7.4.6 Machining contours with SL or OCM cycles

The entire contour is machined with the SL cycles (see "Milling contours with SL cycles ", Page 310) or the OCM cycles (see "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352).

7.5 Point tables

Application

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

Related topics

- Contents of a point table, hiding individual points
  Further information: Programming and Testing User's Manual

Description of function

Coordinates in a point table

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). The coordinates of the spindle axis correspond to the coordinate of the workpiece surface.

The control retracts the tool to the clearance height when traversing between the starting points. Depending on which is greater the control uses either the tool axis coordinate from the cycle call or the value from cycle parameter **Q204 2ND SET-UP CLEARANCE**.

NOTICE

Danger of collision!

If you program a clearance height for individual points in a point table, the control will ignore the value from the cycle parameter **Q204 2ND SET-UP CLEARANCE** for all points!

- Program the function **GLOBAL DEF 125 POSITIONING** so that the control will take into account the clearance height only for the respective point.

Effect with cycles

SL cycles and Cycle 12

The control interprets the points in the point table as an additional datum shift.

Cycles 200 to 208, 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 tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

Cycles 210 to 215

The control interprets the points as an additional datum shift. If you want to use the points defined in the point table as the starting point coordinates, you must program the starting points and the coordinate of the workpiece upper edge (**Q203**) in the respective milling cycle as 0.

You can no longer insert these cycles on the control, but you can edit and run them in existing NC programs.

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 tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

7.5.1 Selecting the point table in the NC program with SEL PATTERN

To select the point table:

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.

- Select **SEL PATTERN**

- Select **File selection**
- > The control opens a window for the file selection.
- Select the desired point table through the file structure
- Confirm your input
- > The control concludes the NC block.

If the point table is not stored in the same directory as the NC program, you must define the complete path name. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: Programming and Testing User's Manual

Example

```
7 SEL PATTERN "TNC:\nc_prog\Positions.PNT"
```

7.5.2 Calling the cycle with a point table

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

CYCL CALL PAT enables the control to execute the point table that you defined last.

To call a cycle in conjunction with a point table:

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.

- Select **CYCL CALL PAT**
- Enter a feed rate

> The control will use this feed rate to traverse between the points of the point table. If you do not enter a feed rate, the control moves the tool at the feed rate last defined.

- Define miscellaneous functions, if necessary
- Confirm your input with the **END** key

Notes

- In the **GLOBAL DEF 125** function you can use the setting **Q435**=**1** to force the control to always move to the 2nd set-up clearance from the cycle during the positioning between the points.
- If you want to move at reduced feed rate when pre-positioning in the tool axis, program the **M103** miscellaneous function.
- With **CYCL CALL PAT** the control runs the point table that you last defined, even if you defined the point table with an NC program that was nested with **CALL PGM**.

7.6 Pattern definition with PATTERN DEF

Application

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

Related topics

- Cycles for pattern definition
  Further information: "Pattern definition cycles", Page 134

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**

To navigate to this function:

Insert NC function ▶ Special functions ▶ Contour/point machining ▶ Pattern ▶ PATTERN DEF

Possible setting	Definition	Further information
POS and **/POS**	Point Definition of up to any 9 machining positions	Page 125
ROW	Row Definition of a single row, straight or rotated	Page 126
PAT	Pattern Definition of a single pattern, straight, rotated or distorted	Page 127
FRAME	Frame Definition of a single frame, straight, rotated or distorted	Page 128
CIRC	Circle Definition of a full circle	Page 130
PITCHCIRC	Pitch circle Definition of a pitch circle	Page 131

Programming PATTERN DEF

To program the **PATTERN DEF** functions:

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.

- Select the desired machining pattern (e.g., **PATTERN DEF CIRC** for a full circle)
- > The control opens the dialog for entering **PATTERN DEF**.
- Enter the required definitions
- Define the machining cycle (e.g., Cycle **200 DRILLING**)
- Call cycle with **CYCL CALL PAT**

While you are programming a machining pattern, you can switch to a different machining pattern in the **Form** column.

Calling PATTERN DEF

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

Further information: "Calling cycles", Page 90

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

Program structure: Machining with PATTERN DEF

```
0 BEGIN SL 2 MM
...
11 PATTERN DEF POS1 (X+25 Y+33.5 Z+0) POS2 (X+15 IY+6.5 Z+0)
12 CYCL DEF 200 DRILLING
...
13 CYCL CALL PAT
```

Notes

Programming note

- Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with **Q345=1**. Then, between the holes, the control always positions the tool to the 2nd set-up clearance that was defined in the cycle.

Operating notes:

- A machining pattern remains active until you define a new one or select a point table with the **SEL PATTERN** function.

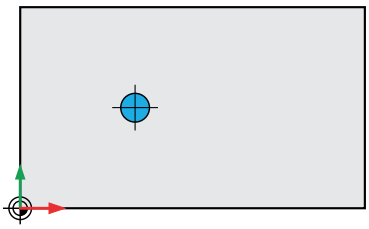
  Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 121
- The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the tool axis position from the cycle call or the value from cycle 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**.
- 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 and Program Run

7.6.1 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.
- You can use the **/POS** syntax element to hide positions that are already defined. The control will then skip these positions.

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**

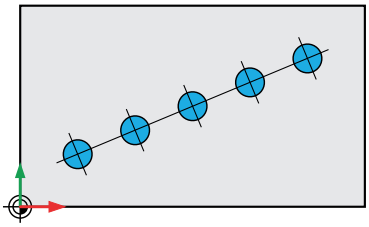
Example

```
11 PATTERN DEF ~
POS1( X+25 Y+33.5 Z+0 ) ~
POS2(  X+15 IY+6.5  Z+0 )
```

7.6.2 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. This value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	Starting point in Y Coordinate of the starting point of the row in the Y axis. This value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	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**

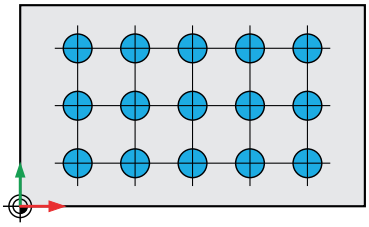
Example

```
11 PATTERN DEF ~
ROW1( X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0 )
```

7.6.3 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**

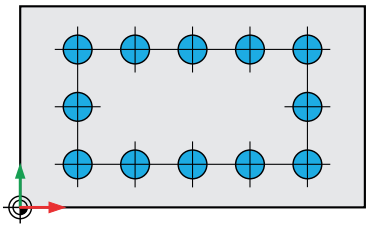
Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate 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 )
```

7.6.4 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**

Help graphic	Parameter
	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**
	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**

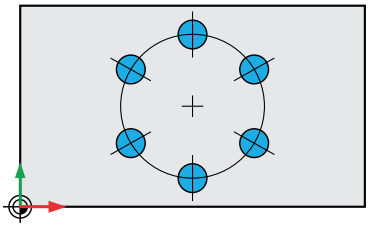
Example

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

7.6.5 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.

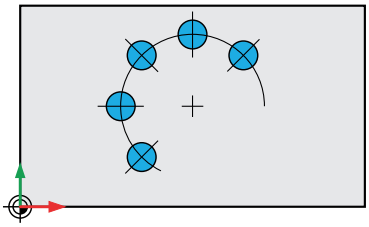
Help graphic	Parameter
	**Bolt-hole circle center X** Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter 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 )
```

7.6.6 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 the selection possibility on the action bar or in the form) 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 )
```

7.6.7 Example: Using cycles in conjunction 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 for drilling, centering, and thread machining", Page 175 and "Milling cycles"

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=+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=+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
20 M30	; End of program run
21 END PGM 1 MM	

7.7 Pattern definition cycles

7.7.1 Overview

The control provides three cycles for machining point patterns:

Cycle		Call	Further information
220	**POLAR PATTERN** ■ Defining a circular pattern ■ Full circle or pitch circle ■ Input of start and end angles	**DEF**-active	Page 136
221	**CARTESIAN PATTERN** ■ Defining a linear pattern ■ Input of an angle of rotation	**DEF**-active	Page 139
224	**DATAMATRIX CODE PATTERN** ■ Converting text to a DataMatrix code to be used as a point pattern ■ Input of position and size	**DEF**-active	Page 142

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: "Point tables", Page 120

Further information: "Pattern definition with PATTERN DEF", Page 123

7.7.2 Cycle 220 POLAR PATTERN

ISO programming
G220

Application

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.

Instead of Cycle **220 POLAR PATTERN**, HEIDENHAIN recommends using the more powerful **PATTERN DEF** function.

Related topics

- Defining a full circle with **PATTERN DEF**
  Further information: "Defining a full circle", Page 130
- Defining a circle segment with **PATTERN DEF**
  Further information: "Defining a pitch circle", Page 131

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 **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

Notes

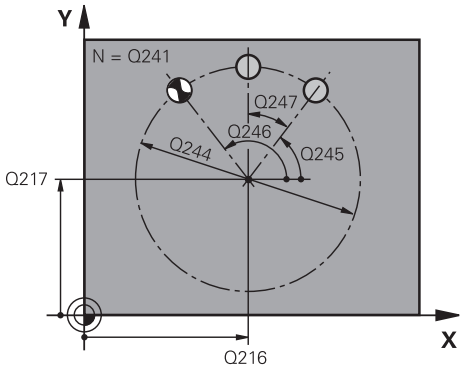
Cycle **220 POLAR PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

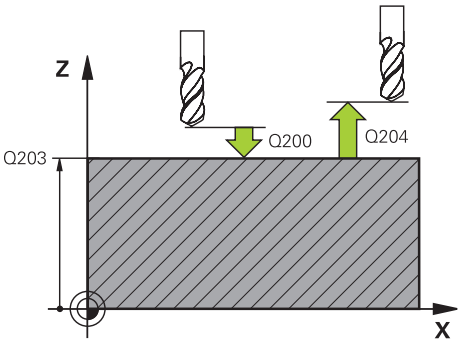
- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

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. This 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. This 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. This 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. This 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. This 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)?** (optional) 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** (optional) 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**

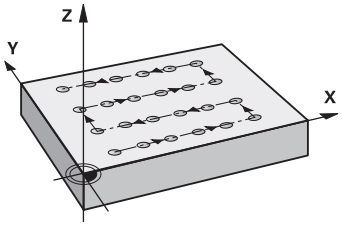
Example

```
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           ;TO CLEARANCE HEIGHT ~
    Q365=+0           ;STRAIGHT LINE
12 CYCL CALL
```

7.7.3 Cycle 221 CARTESIAN PATTERN

ISO programming
G221

Application



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

> Instead of Cycle **221 CARTESIAN PATTERN**, HEIDENHAIN recommends using the more powerful **PATTERN DEF** function.

Related topics

- Defining an individual row with **PATTERN DEF**
  Further information: "Defining a single row", Page 126
- Defining an individual pattern with **PATTERN DEF**
  Further information: "Defining an individual pattern", Page 127

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 row have been completed. The tool is located above the last point of the first row.
5 The tool subsequently moves to the last point on the second row 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 row have been completed.
8 The tool then moves to the starting point of the next row.
9 All subsequent rows are machined in a reciprocating movement.

> If you run this cycle in **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

Notes

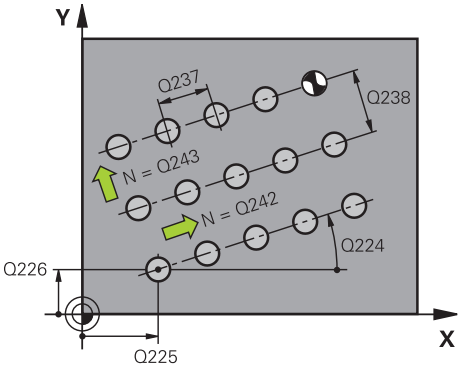
Cycle **221 CARTESIAN PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

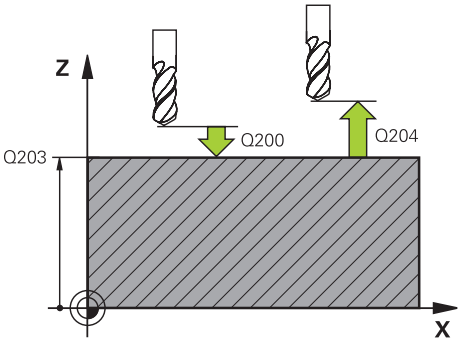
- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

Notes on programming

- If you combine Cycle **221** with one of the machining cycles **200** to **209** or **251** to **267**, then the set-up clearance, the workpiece surface, the 2nd set-up clearance, and the rotary position that you defined in Cycle **221** will be effective for the selected machining cycle.
- 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. This 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. This 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. This value has an absolute effect. Input: **-360.000...+360.000**

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. This 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)?** (optional) 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           ;TO CLEARANCE HEIGHT
12 CYCL CALL
```

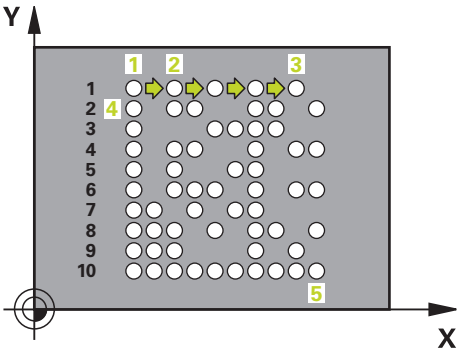
7.7.4 Cycle 224 DATAMATRIX CODE PATTERN

ISO programming
G224

Application

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

Cycle sequence



1 The control automatically positions the tool to the lower left corner of the Data Matrix code.

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 Then, the control moves the tool in the positive direction of the secondary axis to the first starting point **1** in the first row.

3 From this position, the control executes the last defined fixed machining cycle.

4 Then, the control moves the tool in the positive direction of the principal axis to the second starting point **2**of the next machining operation. The tool stops at the 1st set-up clearance.

5 This procedure will be repeated until all machining operations in the first row have been completed. The tool is located above the last point **3** of the first row.

6 Then, the control moves the tool in the negative direction of the principal and secondary axes to the first starting point **4** of the next row.

7 Then, the next points are machined.

8 These steps are repeated until the entire DataMatrix code has been completed. Machining stops in the lower right corner **5**.

9 Finally, the control retracts the tool to the programmed 2nd set-up clearance.

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. There is a danger of collision!

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

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Cycle **224** is DEF-active. In addition, Cycle **224** automatically calls the last defined machining cycle.
- If you select the pattern size in **Q458**, the DataMatrix code limits the pattern size to the overall dimension in **Q459**.

  With the symbol size you define the number of rows and columns. The rows and columns are then integrated in the pattern size. The following examples show you two different situations:

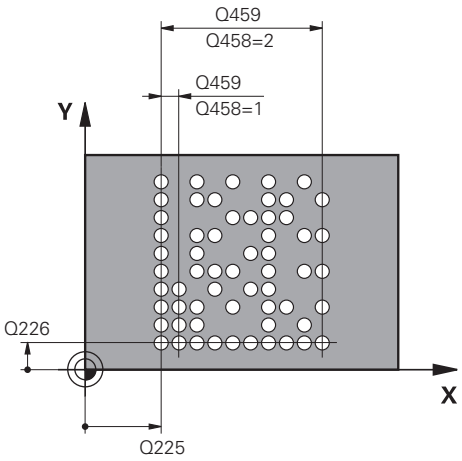
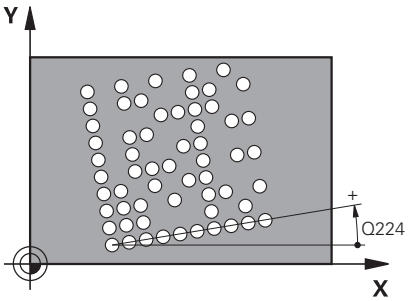
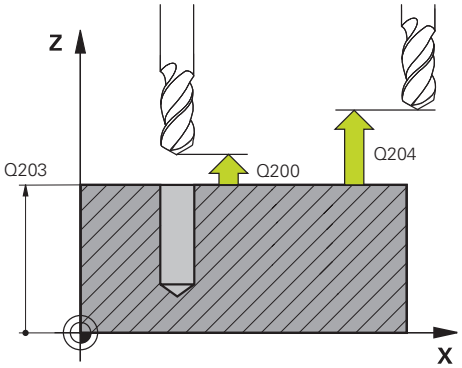
  Example 1:
  - **Q458 SIZE SELECTION**: **2**
  - **Q459 SIZE**: **10**
  - **Q661 SYMBOL SIZE**: **5** (18 rows and 18 columns)

    The control integrates the 18 rows and 18 columns in a DataMatrix code with a side length of 10 mm.

  Example 2:
  - **Q458 SIZE SELECTION**: **2**
  - **Q459 SIZE**: **20**
  - **Q661 SYMBOL SIZE**: **5** (18 rows and 18 columns)

    The control integrates the 18 rows and 18 columns in a DataMatrix code with a side length of 20 mm.

Cycle parameters

Help graphic	Parameter
	**Q225 Starting point in 1st axis?** Coordinate in the main axis. The starting point depends on **Q367 CODE POSITION** (e.g., top right for **Q367=4**). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate in the secondary axis. The starting point depends on **Q367 CODE POSITION** (e.g., top right for **Q367=4**). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	QS501 Text input? Enter the text to be converted within quotation marks. Variables can be assigned. **Further information:** "Outputting variable texts in DataMatrix codes", Page 146 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. This 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. This 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**)
	Q661 Symbol size: Rows * Columns? (optional) Number of rows and columns of the code. Selection possible with a selection menu (e.g., 2=12*12) **0**: Automatic sizing as square. The control generates the code in the minimum size necessary. **1 - 24**: Square **25 - 30**: Rectangular Input: **0...30**
	Q367 Reference for DataMatrix code (0-9)? (optional) Position of **Q225 STARTNG PNT 1ST AXIS** and **Q226 STARTNG PNT 2ND AXIS**. **Reference** ■ 0/1: Bottom left ■ 2: Bottom center ■ 3: Bottom right ■ 4: Top right ■ 5: Top center ■ 6: Top left ■ 7: Center left ■ 8: Center ■ 9: Center right Input: **0...9**

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 ~
    Q661=+0          ;AUTOMATICALLY SQUARE ~
    Q367=+0          ;CODE POSITION
12 CYCL CALL
```

Permitted special characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers: **!#$'()*+,-./:;<=>?@[]_**

- 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., **%%**)
- Umlauts are not possible (ä/ö/ü)

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
%time00	DD.MM.YYYY hh:mm:ss
%time01	D.MM.YYYY h:mm:ss
%time02	D.MM.YYYY h:mm
%time03	D.MM.YY h:mm
%time04	YYYY-MM-DD hh:mm:ss
%time05	YYYY-MM-DD hh:mm
%time06	YYYY-MM-DD h:mm
%time07	YY-MM-DD h:mm
%time08	DD.MM.YYYY
%time09	D.MM.YYYY
%time10	D.MM.YY
%time11	YYYY-MM-DD
%time12	YY-MM-DD
%time13	hh:mm:ss
%time14	h:mm:ss
%time15	h:mm
%time99	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
%main0	Full path of the active NC program	**TNC:\MILL.h**
%main1	Directory path of the active NC program	**TNC:**
%main2	Name of the active NC program	**MILL**
%main3	File type of the active NC program	**.H**
%prog0	Full path of the called NC program	**TNC:\HOUSE.h**
%prog1	Directory path of the called NC program	**TNC:**
%prog2	Name of the called NC program	**HOUSE**
%prog3	File type of the called NC program	**.H**

Count values

You can convert the current counter reading into a DataMatrix code. The current counter reading is displayed during **Program Run** on the **PGM** tab of the **Status** workspace.

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:

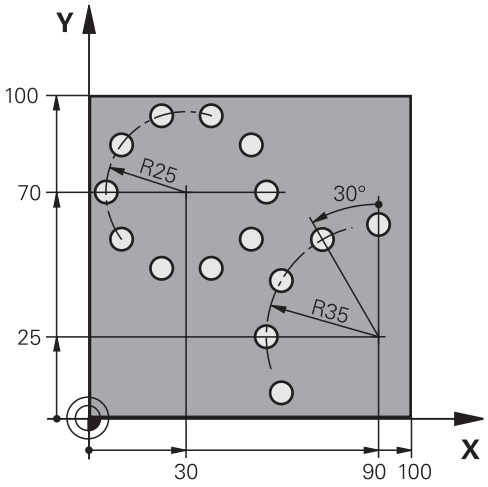
- Programming: **%count9**
- Current count value: 3
- Result: 000000003

Operating information

- During simulation, the control only simulates the counter reading you specified directly in the NC program. The counter reading from the **Status** workspace of the **Program Run** operating mode is ignored.

7.7.5 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 ;TOOL TIP	
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 ;TO CLEARANCE HEIGHT ~	
Q365=+0 ;STRAIGHT LINE	

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 ;TO CLEARANCE HEIGHT ~	
Q365=+0 ;STRAIGHT LINE	
8 L Z+100 R0 FMAX	; Retract the tool
9 M30	; End of program run
10 END PGM 200 MM	

7.8 OCM cycles for figure definition

7.8.1 Overview

OCM figures

Cycle		Call	Further information
1271	**OCM RECTANGLE** (#167 / #1-02-1) ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners	**DEF**-active	Page 153
1272	**OCM CIRCLE** (#167 / #1-02-1) ■ Definition of a circle ■ Input of the circle diameter	**DEF**-active	Page 157
1273	**OCM SLOT / RIDGE** (#167 / #1-02-1) ■ Definition of a slot or ridge ■ Input of the width and the length	**DEF**-active	Page 160
1274	**OCM CIRCULAR SLOT** (#167 / #1-02-1) ■ Definition of a circular slot ■ Input of the width, the pitch circle, and the number of repeats	**DEF**-active	Page 163
1278	**OCM POLYGON** (#167 / #1-02-1) ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners	**DEF**-active	Page 166
1281	**OCM RECTANGLE BOUNDARY** (#167 / #1-02-1) ■ Definition of a bounding rectangle	**DEF**-active	Page 170
1282	**OCM CIRCLE BOUNDARY** (#167 / #1-02-1) ■ Definition of a bounding circle	**DEF**-active	Page 172

7.8.2 Fundamentals

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

These figure cycles offer the following benefits:

- You can conveniently program the figures and machining data without the need to program an individual path contour.
- Frequently needed figures can be reused.
- If you want 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

Related topics

- OCM cycles

  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352

Requirement

- Opt. Contour Milling (#167 / #1-02-1) software option

Description of function

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 provides the following cycles for figure definition:

- **1271 OCM RECTANGLE**, see Page 153
- **1272 OCM CIRCLE**, see Page 157
- **1273 OCM SLOT / RIDGE**, see Page 160
- **1274 OCM CIRCULAR SLOT**, see Page 163
- **1278 OCM POLYGON**, see Page 166

The control provides the following cycles for figure boundary definition:

- **1281 OCM RECTANGLE BOUNDARY**, see Page 170
- **1282 OCM CIRCLE BOUNDARY**, see Page 172

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
1274 OCM CIRCULAR SLOT	Q219 SLOT WIDTH
1278 OCM POLYGON	Q571 REF-CIRCLE DIAMETER

You can define the following tolerances:

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

You can enter nominal dimensions with the following tolerances:

Combination	Example	Manufacturing dimension
a+-b	10+-0.5	10.0
a-+b	10-+0.5	10.0
a-b+c	10-0.1+0.5	10.2
a+b-c	10+0.1-0.5	9.8
a+b+c	10+0.1+0.5	10.3
a-b-c	10-0.1-0.5	9.7
a+b	10+0.5	10.25
a-b	10-0.5	9.75

Proceed as follows:

- Start the cycle definition
- Define the cycle parameters
- Select **NAME** in the action bar
- Enter a nominal dimension including tolerance

- The control produces the workpiece to comply with the mean tolerance value.
- If you program a tolerance that does not comply with the DIN standard or if you indicate tolerances incorrectly when programming nominal dimensions (e.g., by entering blanks), the control aborts execution and displays an error message.
- Ensure correct upper and lower case when entering the DIN EN ISO and DIN ISO tolerances. Entering space characters is not allowed.

7.8.3 Cycle 1271 OCM RECTANGLE (#167 / #1-02-1)

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 an island (**Q650=1**), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**. You define the boundary after the shape cycle.
- If necessary, Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273**, if required **OCM FINISHING FLOOR**
- Cycle **274**, if required **OCM FINISHING SIDE**
- Cycle **277**, if required **OCM CHAMFERING**

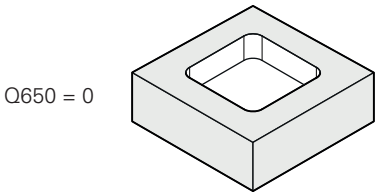
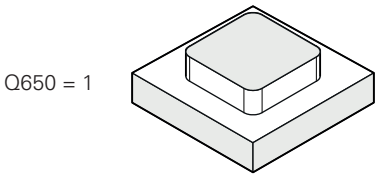
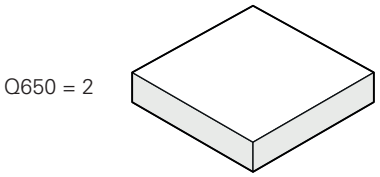
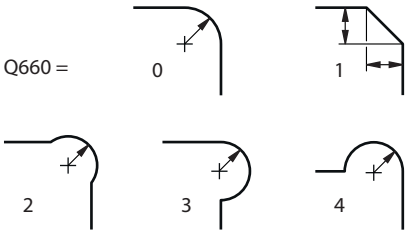
Notes

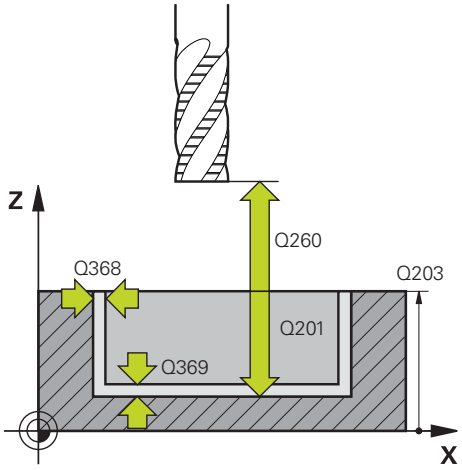
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Cycle **1271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1271** are valid for the OCM machining cycles **272** to **274** and **277**.

Notes on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

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 151 Input: **0...99999.9999**
	**Q219 Second side length?** Length of the second 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 151 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. This 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. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** (or **PREDEF**)
	Q578 Radius factor on inside corners? The tool radius multiplied by **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. Input: **0.05...0.99**

Example

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

7.8.4 Cycle 1272 OCM CIRCLE (#167 / #1-02-1)

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 an island (**Q650=1**), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**. You define the boundary after the shape cycle.
- If necessary, 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

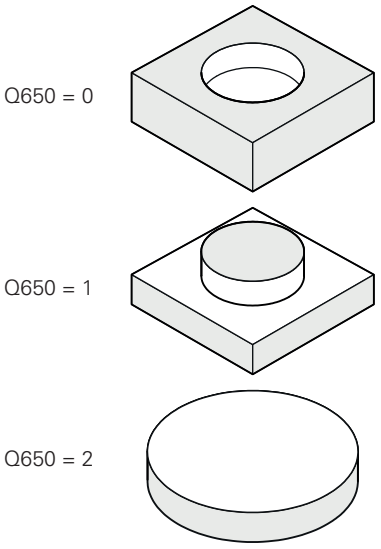
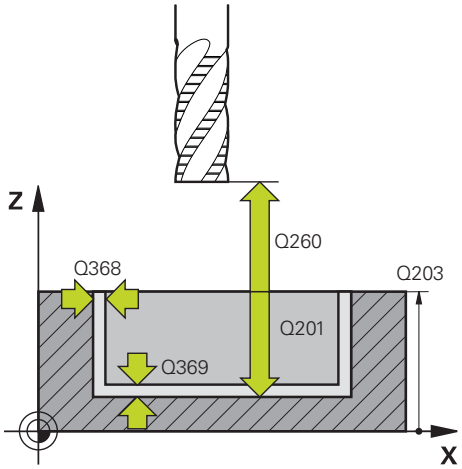
Notes

- This cycle can be executed only 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**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

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 151 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. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** (or **PREDEF**)

Help graphic	Parameter
	Q578 Radius factor on inside corners? The tool radius multiplied by **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1272 OCM CIRCLE ~
    Q650=+0              ;POCKET ~
    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=+50             ;CLEARANCE HEIGHT ~
    Q578=+0.2            ;INSIDE CORNER FACTOR
```

7.8.5 Cycle 1273 OCM SLOT / RIDGE (#167 / #1-02-1)

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 an island (**Q650=1**), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**. You define the boundary after the shape cycle.
- If necessary, 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

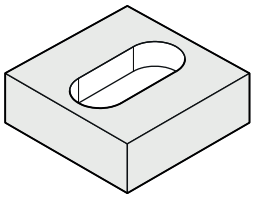
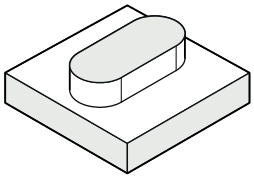
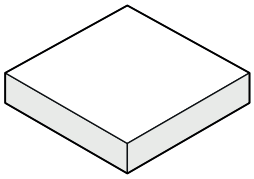
Notes

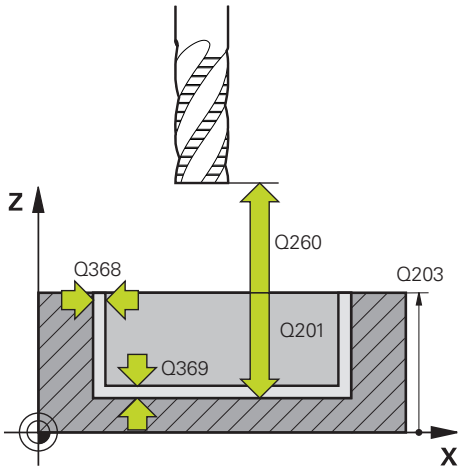
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Cycle **1273** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1273** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

Cycle parameters

Help graphic	Parameter
Q650 = 0  Q650 = 1  Q650 = 2 	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q219 Width of slot? 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. **Further information:** "Tolerances", Page 151 Input: **0...99999.9999**
	Q218 Length of slot? 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. **Further information:** "Tolerances", Page 151 Input: **0...99999.9999**
	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**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. This 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. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999** **Q369 Finishing allowance for floor?** Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999** **Q260 Clearance height?** Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** (or **PREDEF**) **Q578 Radius factor on inside corners?** The tool radius multiplied by **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1273 OCM SLOT / RIDGE ~
    Q650=+0                 ;POCKET ~
    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
```

7.8.6 Cycle 1274 OCM CIRCULAR SLOT (#167 / #1-02-1)

ISO programming
G1274

Application

Use figure cycle **1274 OCM CIRCULAR SLOT** to program a circular slot. Optionally, you can program a tolerance for the slot width.

When using Cycle **1274**, program the cycles in the following sequence:

- Cycle **1274 OCM CIRCULAR SLOT**
- Cycle **272 OCM ROUGHING**
- Cycle **273**, if required **OCM FINISHING FLOOR**
- Cycle **274**, if required **OCM FINISHING SIDE**
- Cycle **277**, if required **OCM CHAMFERING**

Notes

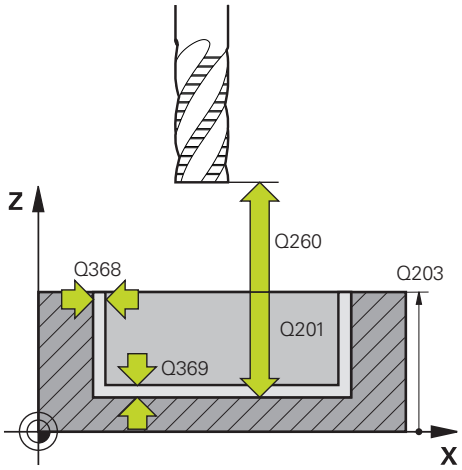
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Cycle **1274** is DEF-active, which means that Cycle **1274** becomes active as soon as it has been defined in the NC program.
- The machining data defined in Cycle **1274** are valid for the OCM machining cycles **272** to **274** and **277**.

Notes on programming

- This cycle requires pre-positioning, which depends on the setting in parameter **Q367 REF. SLOT POSITION**.
- Make sure to define the angle between the starting point and the end point **Q248** in such a way that the contour does not intersect itself. Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	Q219 Width of slot? Slot width This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 151 Input: **0...99999.9999**
	Q375 Pitch circle diameter? The pitch circle diameter is the center line path of the slot. Input: **0...99999.9999**
	Q376 Starting angle? Polar angle of starting point This value has an absolute effect. Input: **-360.000...+360.000**
	Q248 Angular length? The opening angle is the angle between the starting point and the end point of the circular slot. This value has an incremental effect. Input: **0...360**

Help graphic	Parameter
	**Q378 Intermediate stepping angle?** Angle between two machining positions The center of rotation is at the center of the slot. This parameter is effective when the number of machining operations is **Q377>=2**. 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**
	Q367 Ref. for slot pos. (0/1/2/3)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool position = center of the pitch circle **1**: Tool position = center of the left figure arc **2**: Tool position = figure center on center line **3**: Tool position = center of the right figure arc Input: **0**, **1**, **2**, **3**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** (or **PREDEF**)

Help graphic	Parameter
	Q578 Radius factor on inside corners? The tool radius multiplied by **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1274 OCM CIRCULAR SLOT ~
    Q219=+10          ;SLOT WIDTH ~
    Q375=+60          ;PITCH CIRCLE DIAMETR ~
    Q376=+0           ;STARTING ANGLE ~
    Q248=+60          ;ANGULAR LENGTH ~
    Q378=+90          ;STEPPING ANGLE ~
    Q377=+4           ;NR OF REPETITIONS ~
    Q367=+0           ;REF. SLOT POSITION ~
    Q203=+0           ;SURFACE COORDINATE ~
    Q201=-20          ;DEPTH ~
    Q368=+0.1         ;ALLOWANCE FOR SIDE ~
    Q369=+0.1         ;ALLOWANCE FOR FLOOR ~
    Q260=+100         ;CLEARANCE HEIGHT ~
    Q578=+0.2         ;INSIDE CORNER FACTOR
```

7.8.7 Cycle 1278 OCM POLYGON (#167 / #1-02-1)

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 an island (**Q650=1**), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**. You define the boundary after the shape cycle.
- If necessary, 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

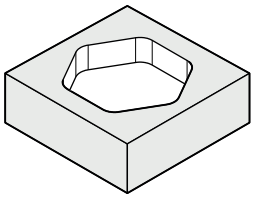
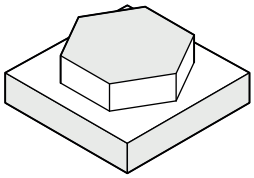
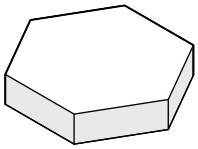
Notes

- This cycle can be executed only 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**.

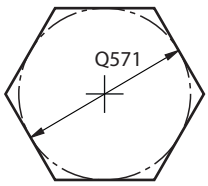
Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

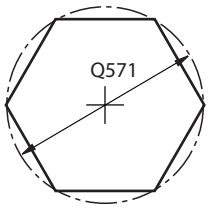
Cycle parameters

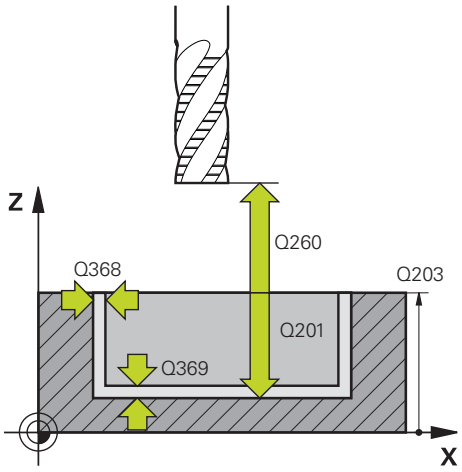
Help graphic	Parameter
Q650 = 0  Q650 = 1  Q650 = 2 	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q573 Inscr.circle/circumcircle (0/1)? Define whether the dimension **Q571** is referenced to the inscribed circle 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 151 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. This value has an absolute effect. Input: **-360.000...+360.000**

Q573 = 0



Q573 = 1



Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** (or **PREDEF**)
	Q578 Radius factor on inside corners? The tool radius multiplied by **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1278 OCM POLYGON ~
    Q650=+0                 ;POCKET ~
    Q573=+0                 ;INSCRIBED CIRCLE IS REFERENCE ~
    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
```

7.8.8 Cycle 1281 OCM RECTANGLE BOUNDARY (#167 / #1-02-1)

ISO programming
G1281

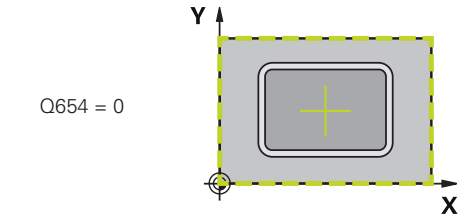
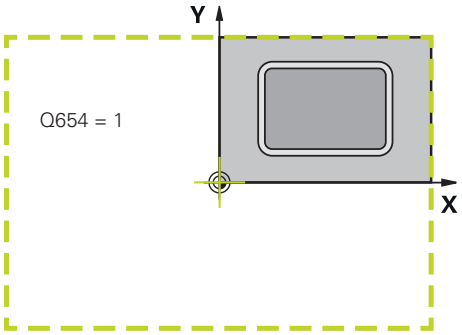
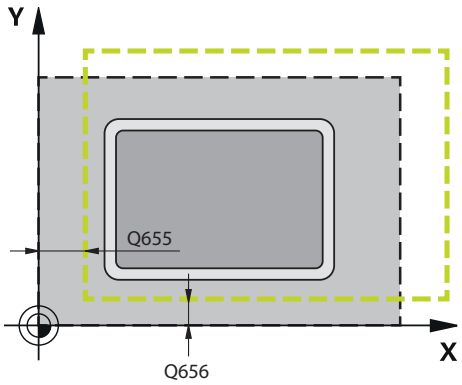
Application

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.

Notes

- This cycle can be executed only 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 **1274** and **1278**.

Cycle parameters

Help graphic	Parameter
	**Q651 Length of major axis?** Length of the first side of the boundary, parallel to the main axis. This value has an incremental effect. Input: **0.001...9999.999**
	**Q652 Length of minor axis?** Length of the second side of the boundary, parallel to the secondary axis. This value has an incremental effect. Input: **0.001...9999.999**
	Q654 Position reference for figure? Specify the position reference for the center: **0**: The center of the boundary is referenced to the center of the contour **1**: The center of the boundary is referenced to the datum Input: **0**, **1**
	**Q655 Shift in major axis?** Shift of the rectangle boundary along the main axis Input: **-999.999...+999.999**
	Q656 Shift in minor axis? Shift of the rectangle boundary along the secondary axis Input: **-999.999...+999.999**

Example

```
11 CYCL DEF 1281 OCM RECTANGLE BOUNDARY ~
    Q651=+50          ;LENGTH 1 ~
    Q652=+50          ;LENGTH 2 ~
    Q654=+0           ;INCREMENTAL TO ISLAND ~
    Q655=+0           ;SHIFT 1 ~
    Q656=+0           ;SHIFT 2
```

7.8.9 Cycle 1282 OCM CIRCLE BOUNDARY (#167 / #1-02-1)

ISO programming
G1282

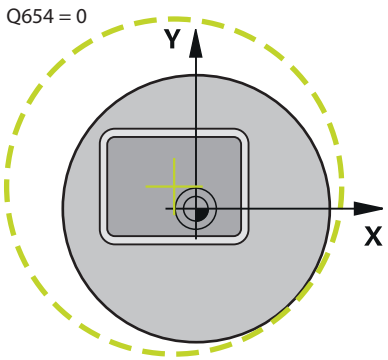
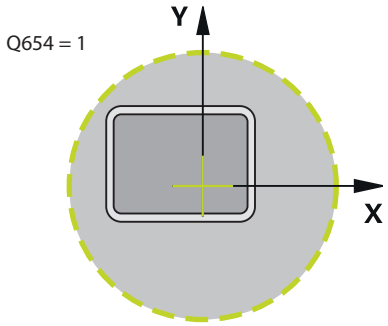
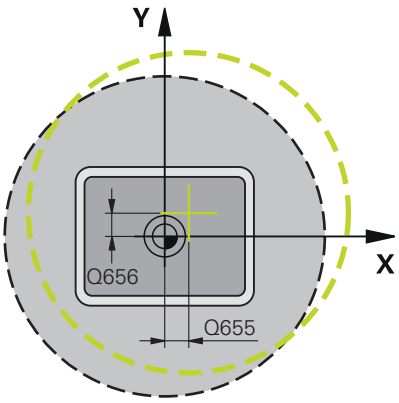
Application

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.

Notes

- This cycle can be executed only 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 **1274** and **1278**.

Cycle parameters

Help graphic	Parameter
	**Q653 Diameter?** Diameter of the circular bounding frame Input: **0.001...9999.999**
	Q654 Position reference for figure? Specify the position reference for the center: **0**: The center of the boundary is referenced to the center of the contour **1**: The center of the boundary is referenced to the datum Input: **0**, **1**
	**Q655 Shift in major axis?** Shift of the rectangle boundary along the main axis Input: **-999.999...+999.999**
	Q656 Shift in minor axis? Shift of the rectangle boundary along the secondary axis Input: **-999.999...+999.999**
	

Example

```
11 CYCL DEF 1282 OCM CIRCLE BOUNDARY ~
    Q653=+50          ;DIAMETER ~
    Q654=+0           ;INCREMENTAL TO ISLAND ~
    Q655=+0           ;SHIFT 1 ~
    Q656=+0           ;SHIFT 2
```

8

Cycles for drilling, centering, and thread machining

8.1 Overview

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

Drilling

Cycle		Call	Further information
200	**DRILLING** ■ Basic hole ■ Input of the dwell time at top and bottom ■ Depth reference selectable	**CALL**-active	Page 179
201	**REAMING** ■ Reaming a hole ■ Input of the dwell time at bottom	**CALL**-active	Page 183
202	**BORING** ■ Boring a hole ■ Input of the retraction feed rate ■ Input of the dwell time at bottom ■ Input of the retracting movement	**CALL**-active	Page 185
203	**UNIVERSAL DRILLING** ■ Degression – hole with decreasing infeed ■ Input of the dwell time at top and bottom ■ Input of chip breaking behavior ■ Depth reference selectable	**CALL**-active	Page 189
205	**UNIVERSAL PECKING** ■ Degression – hole with decreasing infeed ■ Input of chip breaking behavior ■ Input of a deepened starting point ■ Input of an advanced stop distance	**CALL**-active	Page 194
208	**BORE MILLING** ■ Milling of a hole ■ Input of a pre-drill diameter ■ Climb or up-cut milling selectable	**CALL**-active	Page 201
241	**SINGLE-LIP D.H.DRLNG** ■ Drilling with single-lip deep hole drill ■ Deepened starting point ■ Direction of rotation and rotational speed for moving into and retracting from the hole ■ Input of the dwell depth	**CALL**-active	Page 206

Countersinking and centering

Cycle		Call	Further information
204	**BACK BORING** ■ Machining a counterbore on the underside of the workpiece ■ Input of the dwell time ■ Input of the retracting movement	**CALL**-active	Page 214

Cycle		Call	Further information
240	**CENTERING** ■ Drilling a center hole ■ Input of the centering diameter or depth ■ Input of the dwell time at bottom	**CALL**-active	Page 218

Tapping

Cycle		Call	Further information
18	**THREAD CUTTING** ■ With controlled spindle ■ Spindle stops at the bottom of the hole	**CALL**-active	Page 221
206	**TAPPING** ■ With a floating tap holder ■ Input of the dwell time at bottom	**CALL**-active	Page 224
207	**RIGID TAPPING** ■ Without a floating tap holder ■ Input of the dwell time at bottom	**CALL**-active	Page 227
209	**TAPPING W/ CHIP BRKG** ■ Without a floating tap holder ■ Input of chip breaking behavior	**CALL**-active	Page 231

Thread milling

Cycle		Call	Further information
262	**THREAD MILLING** ■ Milling a thread into pre-drilled material	**CALL**-active	Page 237
263	**THREAD MLLNG/CNTSNKG** ■ Milling a thread into pre-drilled material ■ Machining a countersunk chamfer	**CALL**-active	Page 241
264	**THREAD DRILLNG/MLLNG** ■ Drilling into solid material ■ Milling a thread	**CALL**-active	Page 246
265	**HEL. THREAD DRLG/MLG** ■ Milling a thread into solid material	**CALL**-active	Page 251
267	**OUTSIDE THREAD MLLNG** ■ Milling an external thread ■ Machining a countersunk chamfer	**CALL**-active	Page 255

8.2 Conditional stops in drilling and threading operations

If your machine has an override controller, you can activate conditional stops during program run. If you activate conditional stops with the **In cycle call** selection, the control interrupts at the following breakpoints:

The control stops before the first infeed. If you defined a recessed starting point, the control will stop the movement before moving to the recessed starting point.

Further information: User's Manual for Setup and Program Run

8.3 Drilling

8.3.1 Cycle 200 DRILLING

ISO programming
G200

Application

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

Related topics

- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 189
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 194
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 206

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 At the end, 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! There is a danger of collision!

- 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

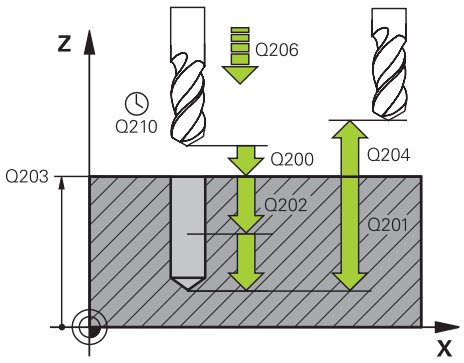
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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. This 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? (optional) 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)? (optional) 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          ;TOOL TIP
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
14 L  X+80  Y+50 FMAX M99
```

8.3.2 Cycle 201 REAMING

ISO programming
G201

Application

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! There is a danger of collision!

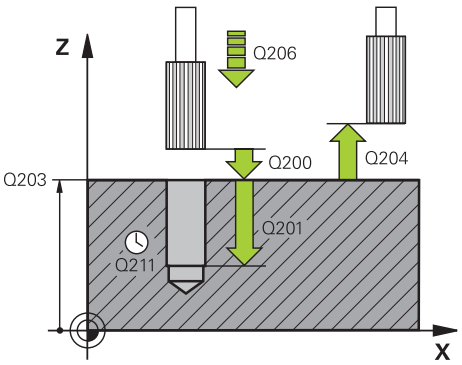
- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- 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. This 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
```

8.3.3 Cycle 202 REAMING

ISO programming
G202

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.

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! There is a danger of collision!

- 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 **MDI** application in the **Manual** 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!

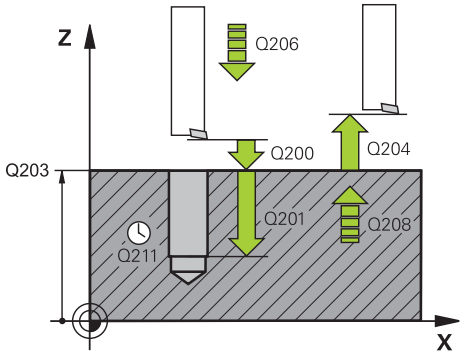
- Use **M137** to deactivate **M136** before the cycle start

- This cycle can be executed only 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. This 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? (optional) Angle to which the control positions the tool before retracting it. This value has an absolute effect. Input: **0...360**

Help graphic	Parameter
	Q357 Safety clearance to the side? (optional) 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 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=+9999       ;RETRACTION FEED RATE ~
    Q203=+0          ;SURFACE COORDINATE ~
    Q204=+50         ;2ND SET-UP CLEARANCE ~
    Q214=+0          ;DO NOT RETRACT ~
    Q336=+0          ;ANGLE OF SPINDLE ~
    Q357=+0.2        ;CLEARANCE TO SIDE
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
```

8.3.4 Cycle 203 UNIVERSAL DRILLING

ISO programming
G203

Application

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.

Related topics

- Cycle **200 DRILLING** for simple holes
  Further information: "Cycle 200 DRILLING", Page 179
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 194
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 206

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 breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and, if necessary, remains there for the **DWELL TIME AT TOP Q210**
6 This sequence will be repeated until the **DEPTH Q201** is reached.
7 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking, without decrement:

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6 If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**
7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
9 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking, with decrement

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

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! There is a danger of collision!

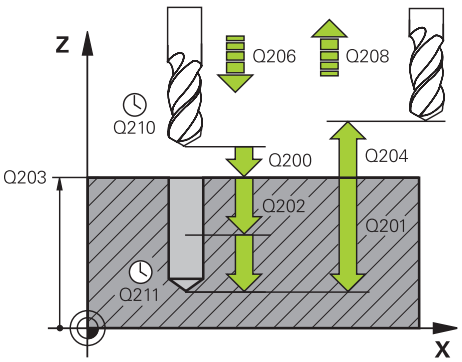
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- 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 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**)

Help graphic	Parameter
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This 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 PLUNGING 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**
	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? (optional) 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)? (optional) 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              ;TOOL TIP
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
```

8.3.5 Cycle 205 UNIVERSAL PECKING

ISO programming
G205

Application

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 199

Related topics

- Cycle **200 DRILLING** for simple holes
  Further information: "Cycle 200 DRILLING", Page 179
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 189
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 206

Cycle sequence

1. The control positions the tool in the tool axis at **FMAX** to the entered **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.
2. If you program a recessed starting point in **Q379**, the control moves at the positioning feed rate **Q253 F PRE-POSITIONING** to the set-up clearance above the recessed 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! There is a danger of collision!

- 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

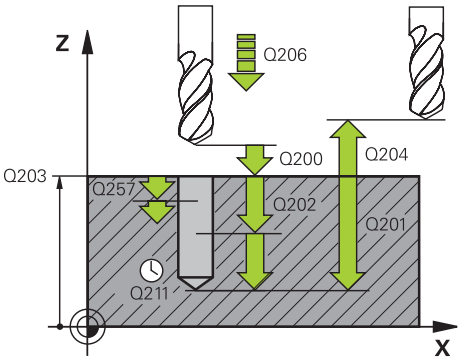
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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. This 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
	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**
	Q259 Lower advanced stop distance? Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after the last chip removal. This value has an incremental effect. Input: **0...99999.9999**
	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? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999** (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**)
	Q379 Deepened starting point? (optional) 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? (optional) Defines the tool traversing speed when positioning from **Q200 SET-UP CLEARANCE** to **Q379 STARTING POINT** (not equal to 0). Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q208 Feed rate for retraction? (optional) Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q395 Diameter as reference (0/1)? (optional) 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? (optional) 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**

```
11 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              ;TOOL TIP ~
    Q373=+0              ;FEED AFTER REMOVAL
7  CYCL CALL
```

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
9 M30	; End of program run
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 **Q201 DEPTH** 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
9 M30	; End of program run
10 END PGM 205 MM	

8.3.6 Cycle 208 BORE MILLING

ISO programming
G208

Application

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:

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

You can enter nominal dimensions with the following tolerances:

Combination	Example	Manufacturing dimension
a+-b	**10+-0.5**	**10.0**
a-+b	**10-+0.5**	**10.0**
a-b+c	**10-0.1+0.5**	**10.2**
a+b-c	**10+0.1-0.5**	**9.8**
a+b+c	**10+0.1+0.5**	**10.3**
a-b-c	**10-0.1-0.5**	**9.7**
a+b	**10+0.5**	**10.25**
a-b	**10-0.5**	**9.75**

Proceed as follows:

- Start the cycle definition
- Define the cycle parameters
- Select **NAME** in the action bar
- Enter a nominal dimension including tolerance

- The control produces the workpiece to comply with the mean tolerance value.
- If you program a tolerance that does not comply with the DIN standard or if you indicate tolerances incorrectly when programming nominal dimensions (e.g., by entering blanks), the control aborts execution and displays an error message.
- Ensure correct upper and lower case when entering the DIN EN ISO and DIN ISO tolerances. Entering space characters is not allowed.

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! There is a danger of collision!

- 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.

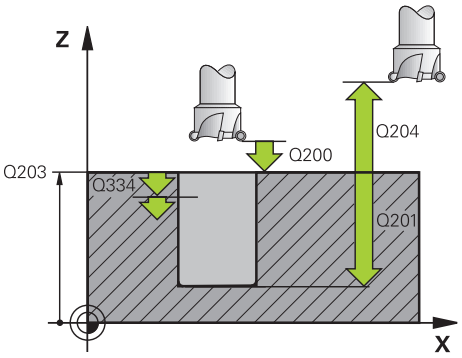
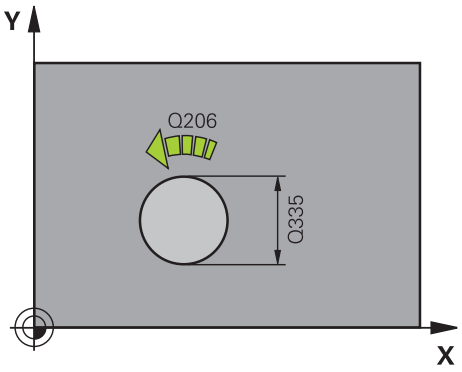
- Specify the maximum possible plunge angle and the corner radius **DR2** in the **ANGLE** column of the **TOOL.T** tool table.
- > The control automatically calculates the max. permissible infeed and changes your entered value accordingly, if necessary.

- This cycle can be executed only 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 so that the bottom of the hole will be as level as possible. The overlap factor is reduced to a minimum.
- 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°). This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This 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. This value has an absolute effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 202 Input: **0...99999.9999**
	Q342 Roughing diameter? (optional) Enter the dimension of the pre-drilled diameter. This value has an absolute effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q351 Direction? Climb=+1, Up-cut=-1 (optional) 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? (optional) 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 MILLING ~
    Q370=+0          ;TOOL PATH OVERLAP
12 CYCL CALL
```

8.3.7 Cycle 241 SINGLE-LIP D.H.DRLNG

ISO programming
G241

Application

Cycle **241 SINGLE-LIP D.H.DRLNG** machines holes with a single-lip deep hole drill. It is possible to enter a recessed starting point. The control performs moving to drilling depth with **M3**. You can change the direction of rotation and the rotational speed for moving into and retracting from the hole.

Related topics

- Cycle **200 DRILLING** for simple holes
  Further information: "Cycle 200 DRILLING", Page 179
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 189
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 194

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above **SURFACE COORDINATE Q203**.
2 Depending on the positioning behavior, the control will either switch on the spindle with the programmed speed at **SET-UP CLEARANCE Q200** or at a certain distance above the coordinate surface.
  Further information: "Positioning behavior during program run with Q379", Page 211
3 If you have defined a recessed starting point **Q379 > 0**, the tool moves over the recessed starting point at **Q253 F PRE-POSITIONING** by the value **Q200 SET-UP CLEARANCE Q379**.
4 The control executes the approach motion depending on the direction of rotation defined in **Q426 DIR. OF SPINDLE ROT.** with a spindle that rotates clockwise or counterclockwise, or is stationary.
5 The tool drills with **M3** and **Q206 FEED RATE FOR PLNGNG** to the drilling depth **Q201** or dwell depth **Q435** or the plunging depth **Q202**:
  - If you have entered **Q435 DWELL DEPTH**, the control reduces the feed rate by **Q401 FEED RATE FACTOR** after the dwell depth has been reached and remains there for **Q211 DWELL TIME AT DEPTH**.
  - If a smaller infeed value has been entered, the control drills to the plunging depth. With each infeed, the plunging depth is reduced by **Q212 DECREMENT**.
6 If programmed, the tool remains at the hole bottom for chip breaking.
7 After the control has reached this position, it will automatically switch off the coolant, set the speed to the value defined in **Q427 ROT.SPEED INFEED/OUT** and, if required, change again the direction of rotation defined in **Q426**.
8 The control positions the tool to the retract position at **Q208 RETRACTION FEED RATE**.
  Further information: "Positioning behavior during program run with Q379", Page 211
9 If programmed, the tool moves to the 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! There is a danger of collision!

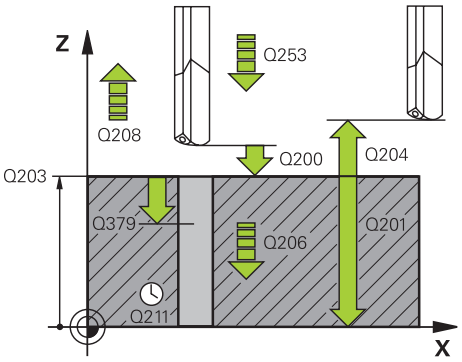
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can be executed only 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. This 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**)
	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**
	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: **1...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 210 Input: **0...999**

Help graphic	Parameter
	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 **Q201 DEPTH**. **"..."**: 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 210 Input: **0...999**
	Q435 Dwell depth? (optional) 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 %? (optional) Factor by which the control reduces the feed rate after reaching **Q435 DWELL DEPTH**. Input: **0.0001...100**
	Q202 Maximum plunging depth? (optional) 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? (optional) Value by which the control decreases **Q202 PLUNGING DEPTH** after each infeed. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q205 Minimum plunging depth? (optional) 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=+99999      ;RETRACTION FEED RATE ~
    Q426=+5          ;SPINDLE STATIONARY ~
    Q427=+50         ;ROT.SPEED INFEED/OUT ~
    Q428=+500        ;ROT. SPEED DRILLING ~
    Q429=+8          ;COOLANT ON ~
    Q430=+9          ;COOLANT OFF ~
    Q435=+0          ;DWELL DEPTH ~
    Q401=+100        ;FEED RATE FACTOR ~
    Q202=+9999       ;MAX. PLUNGING DEPTH ~
    Q212=+0          ;DECREMENT ~
    Q205=+0          ;MIN. PLUNGING DEPTH
12 CYCL CALL
```

User macro

User macros are separate NC programs.

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 NC programs.

Macros work in the same manner as NC programs that are called (e.g., with the NC function **CALL PGM**). 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: Programming and Testing User's Manual

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 **Start** LBL
3 M8	; Switch coolant on
7 CYCL DEF 9.0 DWELL TIME	
8 CYCL DEF 9.1 DWELL3	
9 LBL "Start"	
10 END PGM RET MM	

Positioning behavior during program run with Q379

Especially when working with very long drills (for example, 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.

Starting point for drilling

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

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

The following table lists various examples of how the starting point is calculated:

Drilling start at the recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of 0.2 * Q379	Starting point for drilling
2	2	0	2	0.2*2=0.4	–1.6
2	5	0	2	0.2*5=1	–4
2	10	0	2	0.2*10=2	–8
2	25	0	2	0.2*25=5 (**Q200**=2, 5>2, so the value 2 is used.)	–23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	–98
5	2	0	5	0.2*2=0.4	–1.6
5	5	0	5	0.2*5=1	–4
5	10	0	5	0.2*10=2	–8
5	25	0	5	0.2*25=5	–20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	–95
20	2	0	20	0.2*2=0.4	–1.6
20	5	0	20	0.2*5=1	–4
20	10	0	20	0.2*10=2	–8
20	25	0	20	0.2*25=5	–20
20	100	0	20	0.2*100=20	–80

Chip removal

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

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

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

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

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

8.4 Countersinking and centering

8.4.1 Cycle 204 BACK BORING

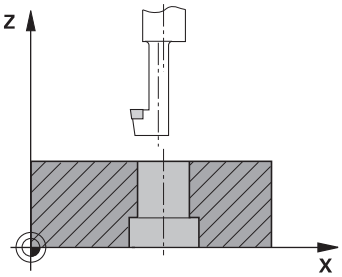
ISO programming
G204

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.

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 At the end, 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 **MDI** application in the **Manual** 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.

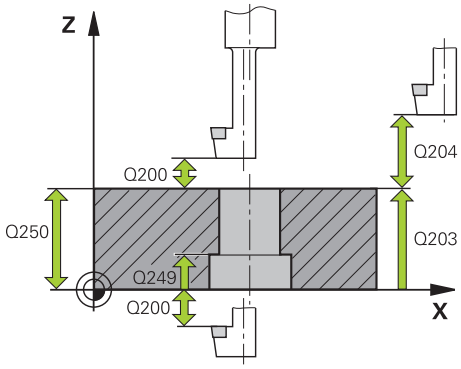
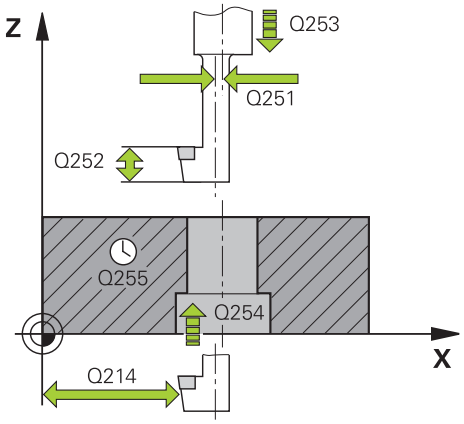
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- When calculating the starting point for boring, the control considers 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...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This 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? (optional) Angle at which the control positions the tool before it is plunged into or retracted from the bore hole This value has an absolute effect. Input: **0...360**

Example

```
11 CYCL DEF 204 BACK BORING ~
    Q200=+2          ;SET-UP CLEARANCE ~
    Q249=+2          ;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          ;DO NOT RETRACT ~
    Q336=+0          ;ANGLE OF SPINDLE
```

8.4.2 Cycle 240 CENTERING

ISO programming
G240

Application

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** at the safety distance **Q200** above the recessed 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 At the end, 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! There is a danger of collision!

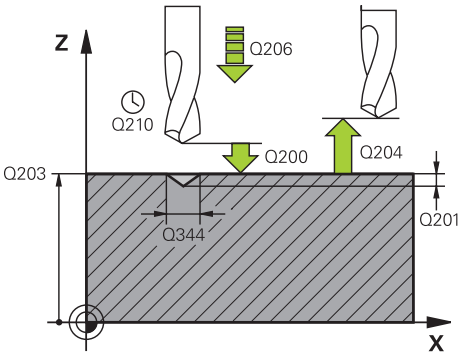
- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- 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. This 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? (optional) **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? (optional) 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

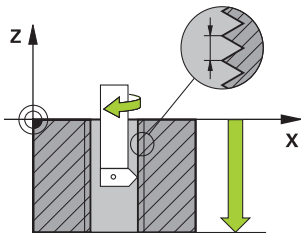
```
11 CYCL DEF 240 CENTERING ~
    Q200=+2          ;SET-UP CLEARANCE ~
    Q343=+1          ;DIAMETER ~
    Q201=-2          ;DEPTH ~
    Q344=-10         ;DIAMETER ~
    Q206=+150        ;FEED RATE FOR PLNGNG ~
    Q211=+0          ;DWELL TIME AT DEPTH ~
    Q203=+0          ;SURFACE COORDINATE ~
    Q204=+50         ;2ND SET-UP CLEARANCE ~
    Q342=+0          ;ROUGHING DIAMETER ~
    Q253=+750        ;F PRE-POSITIONING
```

8.5 Tapping

8.5.1 Cycle 18 THREAD CUTTING

ISO programming
G86

Application



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.

Related topics

- Cycles for Thread Machining
  Further information: "Cycle 206 TAPPING ", Page 224
  Further information: "Cycle 207 RIGID TAPPING ", Page 227
  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 231

Notes

Cycle **18 THREAD CUTTING** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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

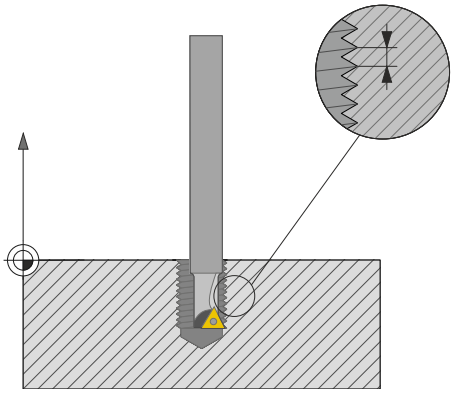
Notes on programming

- 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
	**Total hole depth?** Enter the thread depth relative to the current position. This value has an incremental effect. Input: **-999999999...+999999999** **Thread pitch?** 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: **-99.9999...+99.9999**

Example

```
11 CYCL DEF 18.0 THREAD CUTTING
12 CYCL DEF 18.1 DEPTH-20
13 CYCL DEF 18.2 PITCH+1
```

8.5.2 Cycle 206 TAPPING

ISO programming
G206

Application

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

Related topics

- Cycle **207 RIGID TAPPING** without floating tap holder
  Further information: "Cycle 207 RIGID TAPPING ", Page 227
- Cycle **209 TAPPING W/ CHIP BRKG** without floating tap holder, but optionally with chip breaking
  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 231

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 taps to the total hole depth in one movement.
3 Once the tool has reached this position, 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 the 2nd set-up clearance at **FMAX**.
4 At the set-up clearance, the direction of spindle rotation is reversed once again.

A floating tap holder is required for tapping. It must compensate for 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! There is a danger of collision!

- 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 be executed only 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.

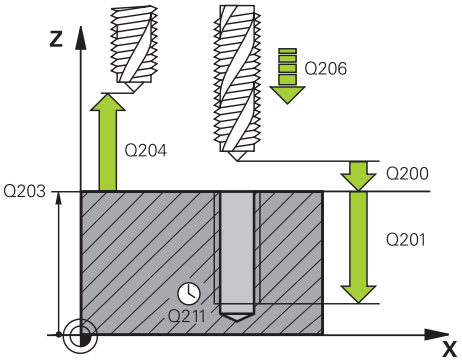
Notes on programming

- 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. This 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 x p

F: Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Retraction with stopped NC program

You can retract a thread-turning tool as follows in stopped state:

- Select **Tool Retract**

- Press the **NC Start** key
- > The tool retracts from the hole and moves to the starting point of machining.
- > The spindle is stopped automatically. The control issues an error message.

- Cancel the NC program with the **Internal stop** button

  or
- Acknowledge the error message and continue with **NC Start**

- **Program Run** operating mode:

  When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.
- **MDI** application:

  When you call a thread cycle, the **Tool Retract** button appears. The button is dimmed until you press **NC stop**.

8.5.3 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.

Related topics

- Cycle **206 TAPPING** with floating tap holder
  Further information: "Cycle 206 TAPPING ", Page 224
- Cycle **209 TAPPING W/ CHIP BRKG** without floating tap holder, but optionally with chip breaking
  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 231

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 taps to the total hole depth in one movement.
3 The direction of spindle rotation is then reversed and the tool is retracted to set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
4 The control stops the spindle rotation at 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

Cycle **207 RIGID TAPPING** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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! There is a danger of collision!

- 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 be executed only 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

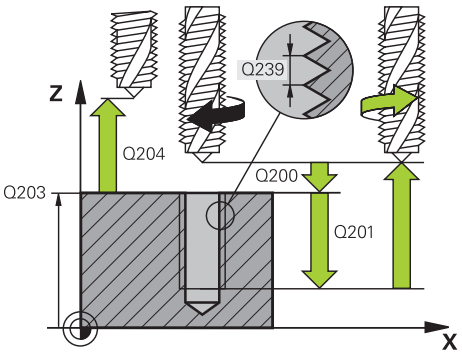
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

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. This 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
```

Retraction with stopped NC program

You can retract a thread-turning tool as follows in stopped state:

- Select **Tool Retract**

- Press the **NC Start** key
- > The tool retracts from the hole and moves to the starting point of machining.
- > The spindle is stopped automatically. The control issues an error message.

- Cancel the NC program with the **Internal stop** button

  or
- Acknowledge the error message and continue with **NC Start**

- **Program Run** operating mode:

  When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.
- **MDI** application:

  When you call a thread cycle, the **Tool Retract** button appears. The button is dimmed until you press **NC stop**.

8.5.4 Cycle 209 TAPPING W/ CHIP BRKG

ISO programming
G209

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 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.

Related topics

- Cycle **206 TAPPING** with floating tap holder
  Further information: "Cycle 206 TAPPING ", Page 224
- Cycle **207 RIGID TAPPING** without floating tap holder
  Further information: "Cycle 207 RIGID TAPPING ", Page 227

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

Cycle **209 TAPPING W/ CHIP BRKG** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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! There is a danger of collision!

- 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 be executed only 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.

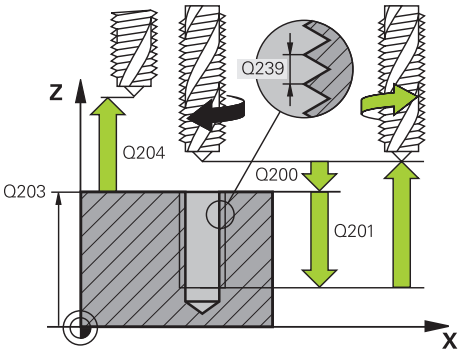
Notes on programming

- 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. This 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 value has an absolute effect. Input: **0...360**

Help graphic	Parameter
	Q403 RPM factor for retraction? (optional) 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
```

Retraction with stopped NC program

You can retract a thread-turning tool as follows in stopped state:

- Select **Tool Retract**

- Press the **NC Start** key
- > The tool retracts from the hole and moves to the starting point of machining.
- > The spindle is stopped automatically. The control issues an error message.

- Cancel the NC program with the **Internal stop** button

  or
- Acknowledge the error message and continue with **NC Start**

- **Program Run** operating mode:
  When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.
- **MDI** application:
  When you call a thread cycle, the **Tool Retract** button appears. The button is dimmed until you press **NC stop**.

8.6 Thread milling

8.6.1 Fundamentals of thread milling

Requirements

- Your machine tool features internal spindle cooling (cooling lubricant at least 30 bar, compressed air supply at least 6 bar)
- 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 milling method **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 program run if the tool breaks
- Switch to the **Manual operation** operating mode in the **MDI** application
- 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.
- When using the thread cycles, **CYLINDER SURFACE** cylinder kinematics must not be active.

8.6.2 Cycle 262 THREAD MILLING

ISO programming
G262

Application

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

Related topics

- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer
  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 241
- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 246
- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer
  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 251
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 255

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! There is a danger of collision!

- 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.

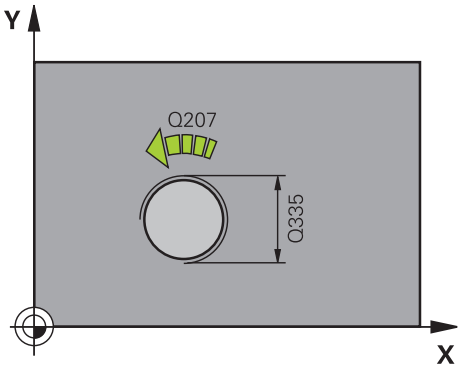
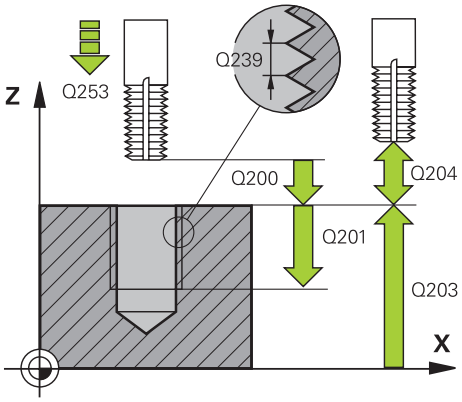
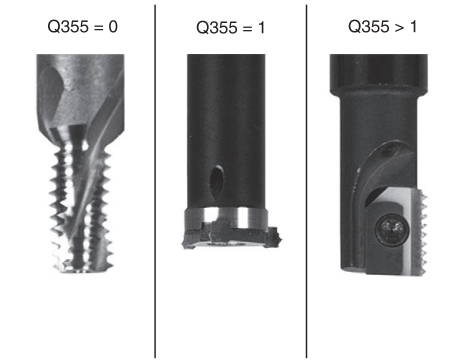
- Ensure sufficient space in the hole!

- This cycle can be executed only 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. This 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? (optional) 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 MILLING ~
    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
```

8.6.3 Cycle 263 THREAD MLLNG/CNTSNKG

ISO programming
G263

Application

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

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material
  Further information: "Cycle 262 THREAD MILLING ", Page 237
- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 246
- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer
  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 251
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 255

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! There is a danger of collision!

- 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

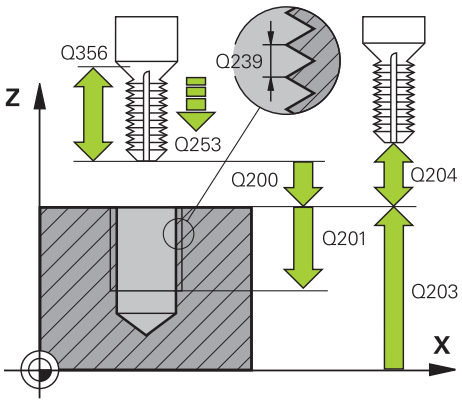
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1 Depth of thread
  2 Countersinking depth
  3 Depth at front

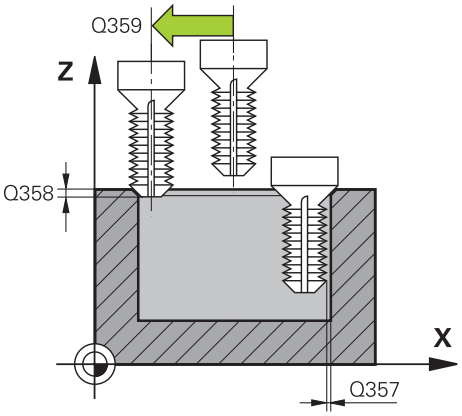
Notes on programming

- 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. This 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? (optional) 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 MILLING ~
    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
```

8.6.4 Cycle 264 THREAD DRILLNG/MLLNG

ISO programming
G264

Application

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

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material
  Further information: "Cycle 262 THREAD MILLING ", Page 237
- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer
  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 241
- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer
  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 251
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 255

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! There is a danger of collision!

- 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

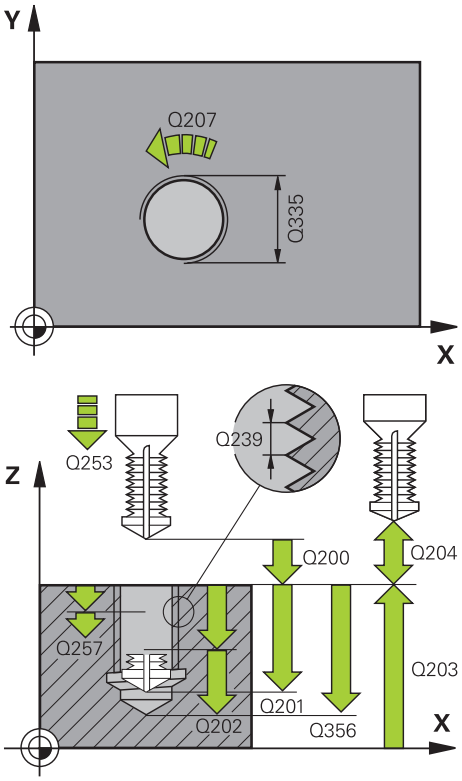
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1 Depth of thread
  2 Countersinking 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
	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? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999** (or **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**
	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. This 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**)
	Q206 Feed rate for plunging? Tool traversing speed in mm/min during plunging 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? (optional) 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 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 MILLING ~
    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
```

8.6.5 Cycle 265 HEL. THREAD DRLG/MLG

ISO programming
G265

Application

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.

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material
  Further information: "Cycle 262 THREAD MILLING ", Page 237
- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer
  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 241
- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 246
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 255

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 pre-positioning to the starting plane for the thread
6 The tool then approaches the nominal thread diameter tangentially in a helical movement
7 The tool moves on a continuous helical downward path until the thread depth value is reached
8 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
9 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 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! There is a danger of collision!

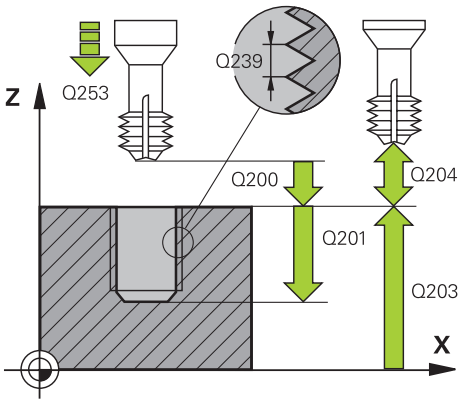
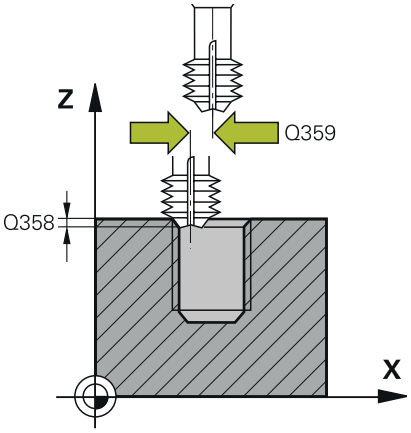
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can be executed only 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. This 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 BEFORE THREADS ~
    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
```

8.6.6 Cycle 267 OUTSIDE THREAD MLLNG

ISO programming
G267

Application

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

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material
  Further information: "Cycle 262 THREAD MILLING ", Page 237
- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer
  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 241
- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 246
- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer
  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 251

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! There is a danger of collision!

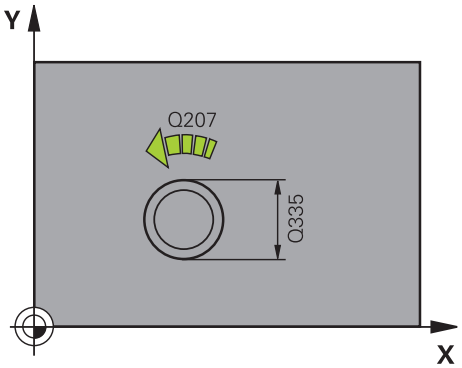
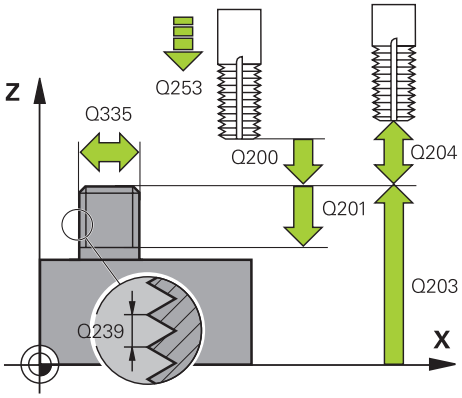
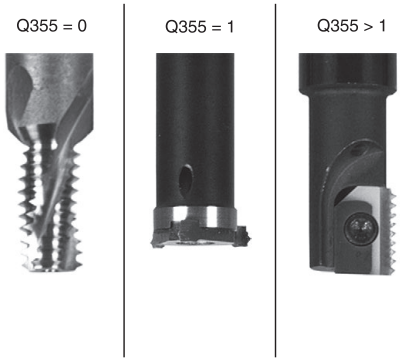
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can be executed only 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**)
	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**

Help graphic	Parameter
	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. This 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? (optional) 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 MILLING ~
    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
```

9

Milling cycles

9.1 Overview

Pocket milling

Cycle		Call	Further information
251	**RECTANGULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	**CALL**-active	Page 264
252	**CIRCULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical or vertical	**CALL**-active	Page 271
253	**SLOT MILLING** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL**-active	Page 277
254	**CIRCULAR SLOT** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL**-active	Page 283

Stud milling

Cycle		Call	Further information
256	**RECTANGULAR STUD** ■ Roughing and finishing cycle ■ Approach position: selectable	**CALL**-active	Page 290
257	**CIRCULAR STUD** ■ Roughing and finishing cycle ■ Input of the start angle ■ Helical infeed starting from the workpiece blank diameter	**CALL**-active	Page 296
258	**POLYGON STUD** ■ Roughing and finishing cycle ■ Helical infeed starting from the workpiece blank diameter	**CALL**-active	Page 301

Milling contours with SL cycles

Cycle		Call	Further information
20	**CONTOUR DATA** ■ Input of machining information	**DEF**-active	Page 313
21	**PILOT DRILLING** ■ Machining a hole for non-center cutting tools	**CALL**-active	Page 315
22	**ROUGH-OUT** ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	**CALL**-active	Page 317
23	**FLOOR FINISHING** ■ Finishing with finishing allowance for the floor from Cycle **20**	**CALL**-active	Page 321

Cycle		Call	Further information
24	**SIDE FINISHING** ■ Finishing with side finishing allowance from Cycle **20**	**CALL**-active	Page 324
270	**CONTOUR TRAIN DATA** ■ Input of contour data for Cycle **25** or **276**	**DEF**-active	Page 328
25	**CONTOUR TRAIN** ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	**CALL**-active	Page 330
275	**TROCHOIDAL SLOT** ■ Machining of open and closed slots using trochoidal milling	**CALL**-active	Page 335
276	**THREE-D CONT. TRAIN** ■ Machining of open and closed contours ■ Detection of residual material ■ 3D contours—additional processing of coordinates from the tool axis	**CALL**-active	Page 341

Milling contours with OCM Cycles

Cycle		Call	Further information
271	**OCM CONTOUR DATA** (#167 / #1-02-1) ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block	**DEF**-active	Page 360
272	**OCM ROUGHING** (#167 / #1-02-1) ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable	**CALL**-active	Page 363
273	**OCM FINISHING FLOOR** (#167 / #1-02-1) ■ Finishing with finishing allowance for the floor from Cycle **271** ■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances)	**CALL**-active	Page 368
274	**OCM FINISHING SIDE** (#167 / #1-02-1) ■ Finishing with side finishing allowance from Cycle **271**	**CALL**-active	Page 371
277	**OCM CHAMFERING** (#167 / #1-02-1) ■ Deburr the edges ■ Consideration of adjacent contours and walls	**CALL**-active	Page 374

Milling planes

Cycle			Further information
232	**FACE MILLING** ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	**CALL**-active	Page 390
233	**FACE MILLING** ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls	**CALL**-active	Page 397

Engraving

Cycle			Further information
225	**ENGRAVING** ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	**CALL**-active	Page 407

9.2 Conditional stops in milling cycles

If your machine has an override controller, you can activate conditional stops during program run. If you activate conditional stops with the **In cycle call** selection, the control interrupts at the following breakpoints:

The control stops before each infeed movement in the tool-axis direction. Depending on whether the infeed starts at the set-up clearance, the 2nd set-up clearance, or the clearance height, the conditional stop will occur at that position.

Exceptions:

Cycle	Meaning
Cycle **225 ENGRAVING**	The control stops the cycle before the first infeed to engrave a character.
Cycle **291 COUPLG.TURNG.INTERP.**	The control stops after the spindle has been coupled. If the spindle is not coupled, there is no conditional stop.
Cycle **292 CONTOUR.TURNG.INTRP.**	The control stops before the first infeed after the spindle has been coupled.
Cycle **286 GEAR HOBBING** and Cycle **287 GEAR SKIVING**	The control stops before positioning the tool at set-up clearance and before any infeed in the diameter direction.

Further information: User's Manual for Setup and Program Run

9.3 Milling pockets

9.3.1 Cycle 251 RECTANGULAR POCKET

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

1 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.
2 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! There is a danger of collision!

- 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 be executed only 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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.

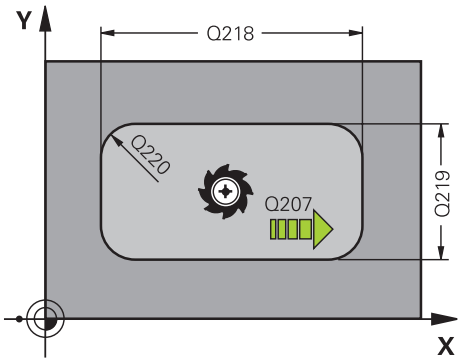
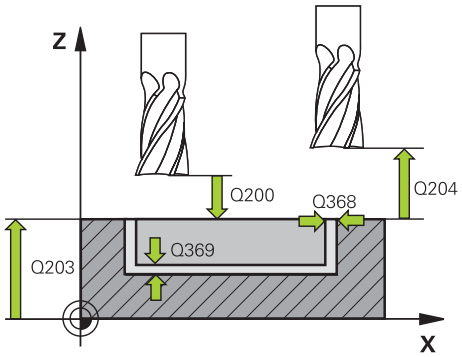
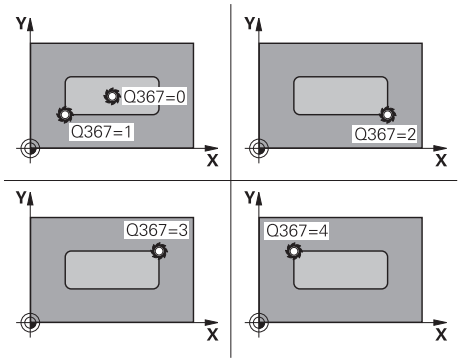
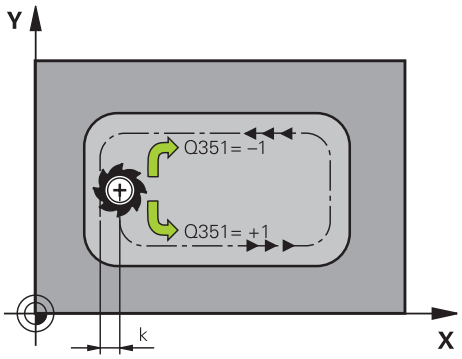
  Further information: "Plunging strategy Q366 with RCUTS", Page 270

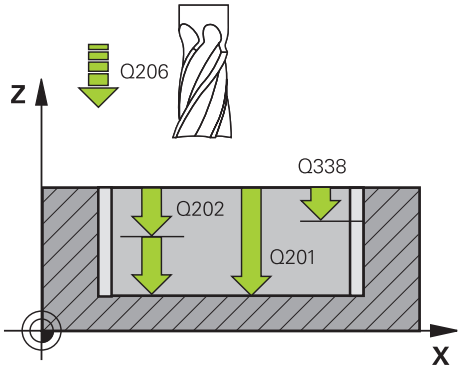
Notes on programming

- If the tool table is inactive, you must always program vertical plunging (**Q366**=0) because a plunging angle cannot be defined.
- 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 executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**

Help graphic	Parameter
	**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 machining plane which remains after roughing. 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. This 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**
	**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 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 in depth which remains after roughing. 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? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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. This 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
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.41** (or **PREDEF**)
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The control plunges vertically, regardless of the plunging angle **ANGLE** defined in the tool table. **1**: Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. If necessary, define the value of the **RCUTS** cutting width in the tool table. **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. 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 **RCUTS** cutting width in the tool table. **PREDEF**: The control uses the value from the GLOBAL DEF block Input: **0**, **1**, **2** (or **PREDEF**) **Further information:** "Plunging strategy Q366 with RCUTS", Page 270
	Q385 Finishing feed rate? (optional) 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)? (optional) 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 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 MILLING ~
    Q201=-20             ;DEPTH ~
    Q202=+5              ;PLUNGING DEPTH ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q338=+0              ;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              ;HELICAL PLUNGE ~
    Q385=+500            ;FINISHING FEED RATE ~
    Q439=+0              ;CENTER POINT PATH
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:

  $Helicalradius = R_{corr} - RCUTS$

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

RCUTS = 0 or undefined

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

Reciprocating plunge Q366 = 2

RCUTS > 0

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

RCUTS = 0 or undefined

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

9.3.2 Cycle 252 CIRCULAR POCKET

ISO programming
G252

Application

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! There is a danger of collision!

- 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 be executed only 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.

- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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 276

Notes on programming

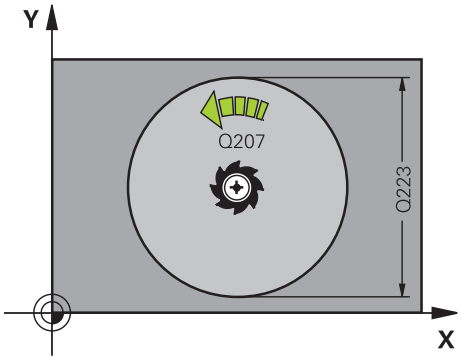
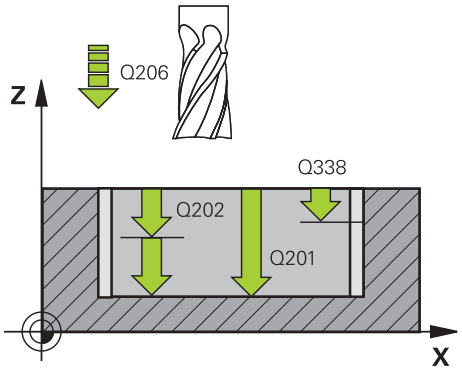
- If the tool table is inactive, you must always program vertical plunging (**Q366**=0) because a plunging angle cannot be defined.
- Pre-position the tool in the working plane to the starting position (circle center) with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter 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.

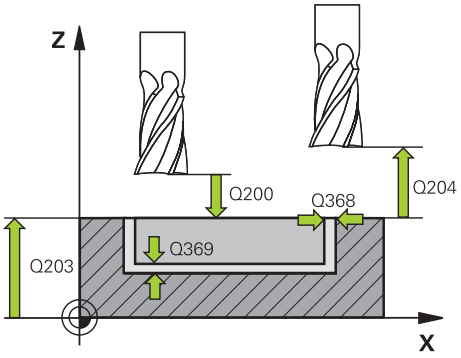
Note regarding machine parameters

- 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 executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**

Help graphic	Parameter
	**Q223 Circle diameter?** Diameter of the finished pocket Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. 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 in depth which remains after roughing. 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? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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**)

Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. This 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 276 **Q385 Finishing feed rate?** (optional) 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)?** (optional) 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                 ;ROUGHING AND FINISHING ~
    Q223=+50                ;CIRCLE DIAMETER ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB MILLING ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;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                 ;HELICAL PLUNGE ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q439=+0                 ;CENTER POINT PATH
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**, the smaller the helical path.
- Formula for calculating the helical radius:

  $Helicalradius = R_{corr} - RCUTS$

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

RCUTS = 0 or undefined

- **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.

9.3.3 Cycle 253 SLOT MILLING

ISO programming
G253

Application

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

1 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
2 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! There is a danger of collision!

- 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! There is a danger of collision!

- 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

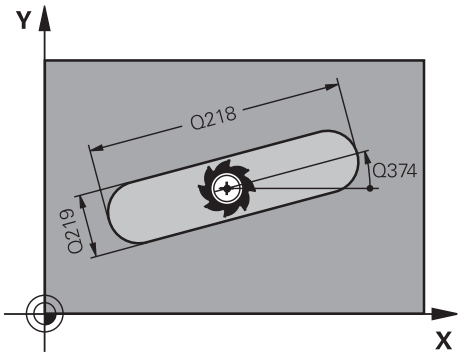
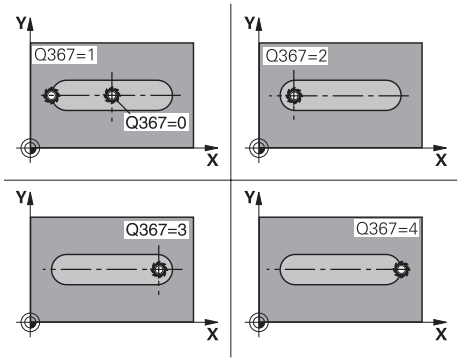
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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.

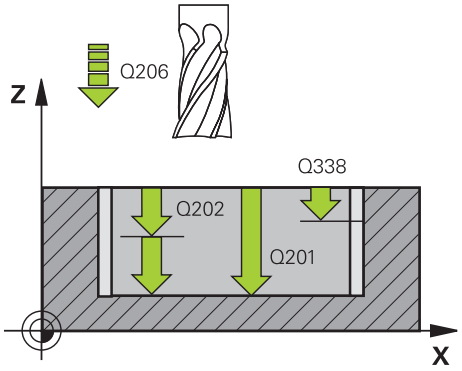
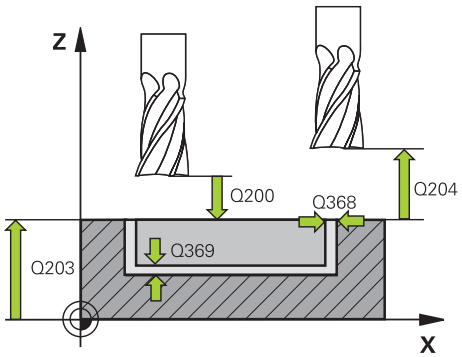
Notes on programming

- If the tool table is inactive, you must always program vertical plunging (**Q366**=0) because a plunging angle cannot be defined.
- 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 executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**

Help graphic	Parameter
	**Q218 Length of slot?** Enter the length of the slot. It is parallel to the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q219 Width of slot? Enter the width of the slot, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. This value has an incremental effect. Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. 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. This 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**
	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**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. 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? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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. This 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
	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. (or **PREDEF**) Input: **0**, **1**, **2**
	Q385 Finishing feed rate? (optional) 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)? (optional) 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 253 SLOT MILLING ~
    Q215=+0                 ;ROUGHING AND FINISHING ~
    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 MILLING ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q366=+2                 ;RECIPROCATING PLUNGE ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q439=+3                 ;ON CONTOUR
12 L  X+50  Y+50 R0 FMAX M99
```

9.3.4 Cycle 254 CIRCULAR SLOT

ISO programming
G254

Application

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

1 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
2 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! There is a danger of collision!

- 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! There is a danger of collision!

- 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.

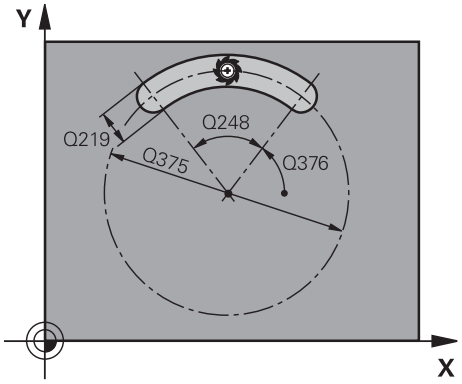
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

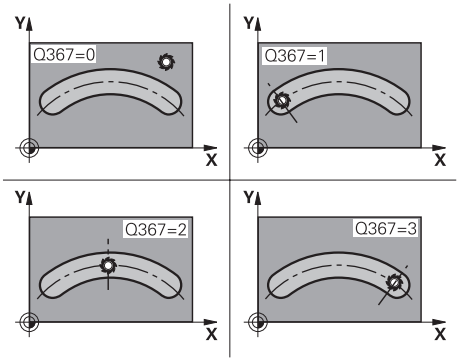
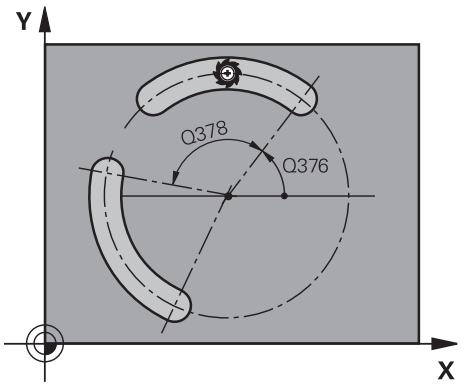
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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.

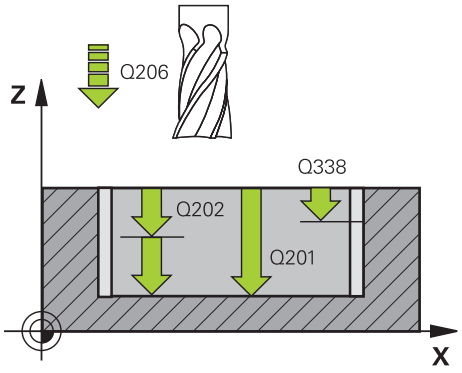
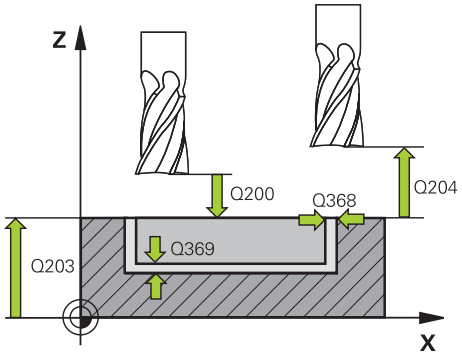
Notes on programming

- If the tool table is inactive, you must always program vertical plunging (**Q366**=0) because a plunging angle cannot be defined.
- 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 executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q219 Width of slot?** Enter the width of the slot, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. This value has an incremental effect. Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q375 Pitch circle diameter? The pitch circle diameter is the center line path of the slot. Input: **0...99999.9999**

Help graphic	Parameter
	**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**. This 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**. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	**Q376 Starting angle?** Polar angle of starting point Input: **-360.000...+360.000**
	Q248 Angular length? The opening angle is the angle between the starting point and the end point of the circular slot. This value has an incremental effect. Input: **0...360**
	Q378 Intermediate stepping angle? Angle between two machining positions 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**

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 in depth which remains after roughing. 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? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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. This 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)?

Help graphic	Parameter
	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? (optional) 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)? (optional) 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              ;ROUGHING AND FINISHING ~
    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 MILLING ~
    Q201=-20             ;DEPTH ~
    Q202=+5              ;PLUNGING DEPTH ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q338=+0              ;INFEED FOR FINISHING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q366=+2              ;RECIPROCATING PLUNGE ~
    Q385=+500            ;FINISHING FEED RATE ~
    Q439=+0              ;CENTER POINT PATH
12 L  X+50  Y+50 R0 FMAX M99
```

9.4 Milling studs

9.4.1 Cycle 256 RECTANGULAR STUD

ISO programming
G256

Application

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! There is a danger of collision!

- 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.

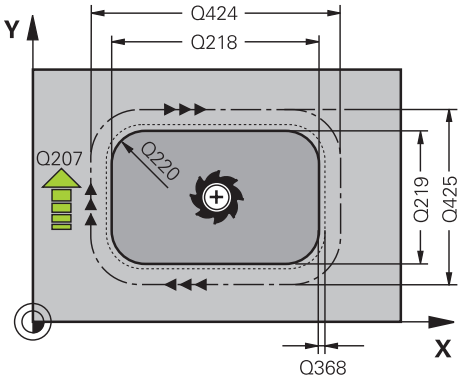
- Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

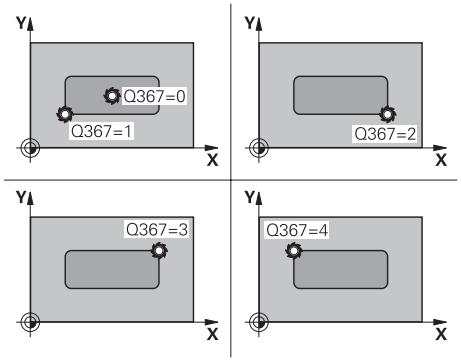
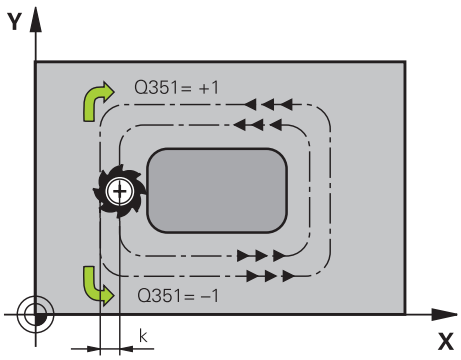
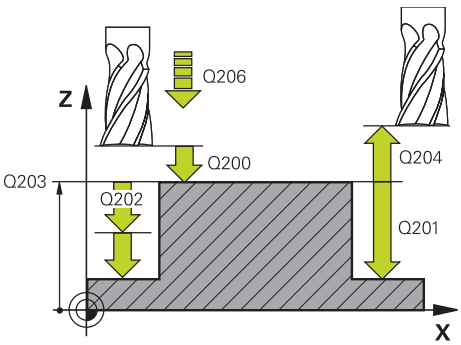
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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 This value has an incremental effect. 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. This value has an incremental effect. 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. This value has an incremental effect. Input: **0...99999.9999**
	Q425 Workpiece blank side length 2? Length of stud blank parallel to the secondary axis of the working plane. This value has an incremental effect. 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 machining plane which remains after roughing. 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. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	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**)
	Q437 Starting position (0...4)? (optional) Specify the approach strategy of the tool: **0**: From the right of the stud (default setting) **1**: Lower left corner **2**: Lower right corner **3**: Upper right corner **4**: Upper left corner If approach marks appear on the stud surface during approach with the setting **Q437**=0, then choose another approach position. Input: **0**, **1**, **2**, **3**, **4**
	Q215 Machining operation (0/1/2)? (optional) Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? (optional) Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? (optional) Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q385 Finishing feed rate? (optional) 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 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 MILLING ~
    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               ;RIGHT ~
    Q215=+1               ;ROUGHING ~
    Q369=+0               ;ALLOWANCE FOR FLOOR ~
    Q338=+0               ;INFEED FOR FINISHING ~
    Q385=+500             ;FEED RATE FOR FINISHING
12 L  X+50  Y+50 R0 FMAX M99
```

9.4.2 Cycle 257 CIRCULAR STUD

ISO programming
G257

Application

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 first 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! There is a danger of collision!

- 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.

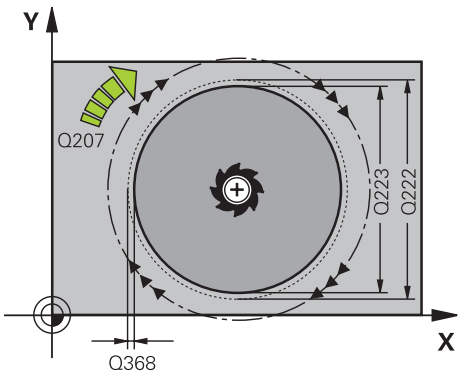
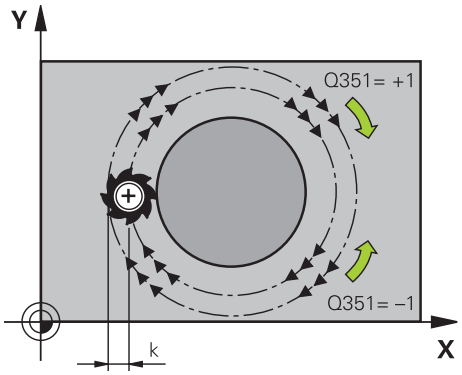
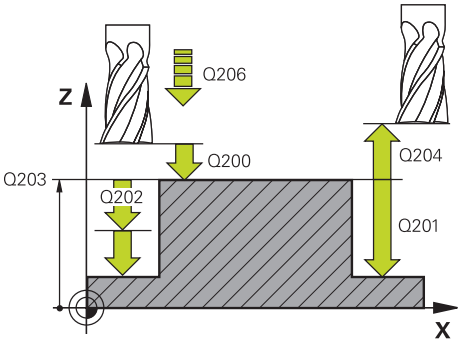
- Check the machining sequence using the graphic simulation.

- This cycle can be executed only 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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 machining plane which remains 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**
	**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. This 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? (optional) Polar angle relative to the stud center, from which the tool approaches the stud. Input: **-1...+359**
	Q215 Machining operation (0/1/2)? (optional) Specify the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? (optional) Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? (optional) Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q385 Finishing feed rate? (optional) 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 MILLING ~
    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               ;ROUGHING ~
    Q369=+0               ;ALLOWANCE FOR FLOOR ~
    Q338=+0               ;INFEED FOR FINISHING ~
    Q385=+500             ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

9.4.3 Cycle 258 POLYGON STUD

ISO programming
G258

Application

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! There is a danger of collision!

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

NOTICE

Danger of collision!

In this cycle, the control performs an automatic approach movement. If there is not enough space, a collision might occur.

- Use **Q224** to specify which angle is used to machine the first corner of the polygon stud. Input range: –360° to +360°
- Depending on the angle of rotation **Q224**, the following amount of space must be left next to the stud: At least tool diameter +2 mm

NOTICE

Danger of collision!

At the end, the control returns the tool to the set-up clearance, or to 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. There is a danger of collision!

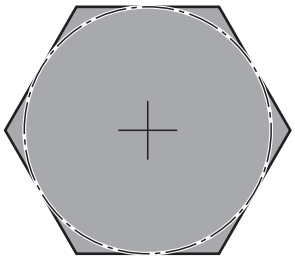
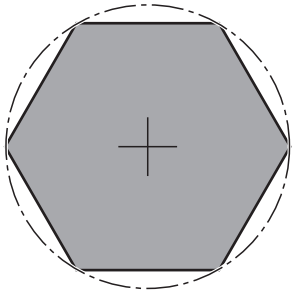
- Control the traversing movements of the machine
- In the **Simulation** workspace of the **Editor** operating mode, check the end position of the tool after the cycle
- After the cycle, program absolute coordinates (no incremental coordinates)

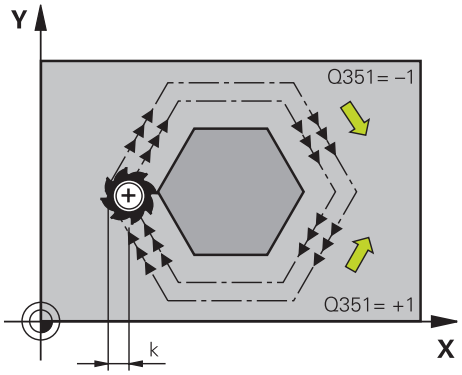
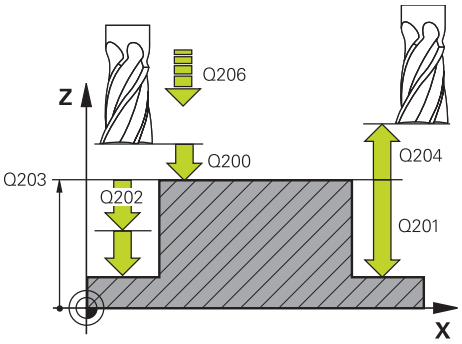
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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	Parameter
Q573 = 0 	**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**
Q573 = 1 	**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. This 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**)

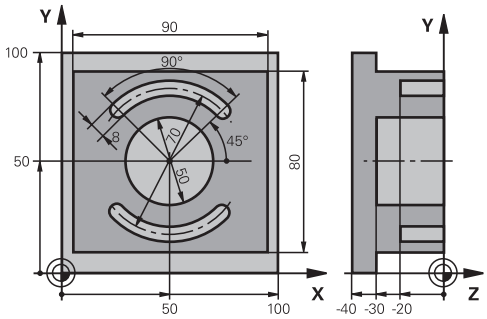
Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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**

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 MILLING ~
    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                 ;ROUGHING AND FINISHING ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q385=+500               ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

9.4.4 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 ;RADIUS / CHAMFER ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB MILLING ~	
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 ;RIGHT ~	
Q215=+0 ;ROUGHING AND FINISHING ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q338=+10 ;INFEED 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 ;ROUGHING AND FINISHING ~	

Q223=+50 ;CIRCLE DIAMETER ~	
Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB MILLING ~	
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 ;HELICAL PLUNGE ~	
Q385=+750 ;FINISHING FEED RATE ~	
Q439=+0 ;CENTER POINT PATH	
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 ;ROUGHING AND FINISHING ~	
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 MILLING ~	
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 ;RECIPROCATING PLUNGE ~	
Q385=+500 ;FINISHING FEED RATE ~	

Q439=+0 ;CENTER POINT PATH	
12 CYCL CALL	; Cycle call for slots
13 L Z+100 R0 FMAX	; Retract the tool
14 M30	; End of program run
15 END PGM C210 MM	

9.5 Milling contours with SL cycles

9.5.1 Fundamentals

Application

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**.

Instead of SL cycles, HEIDENHAIN recommends using the more powerful software option Opt. Contour Milling (#167 / #1-02-1).

Related topics

- Optimized contour milling (#167 / #1-02-1)
  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 352
- Contour call with a simple contour formula **CONTOUR DEF**
  Further information: "Simple contour formula", Page 107
- Contour call with a complex contour formula **SEL CONTOUR**
  Further information: "Complex contour formula", Page 111
- Contour call with cycle **14 CONTOUR**
  Further information: "Cycle 14 CONTOUR ", Page 106

Description of function

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

```
0 BEGIN SL 2 MM
...
12 CYCL DEF 14 CONTOUR
...
13 CYCL DEF 20 CONTOUR DATA
...
16 CYCL DEF 21 PILOT DRILLING
...
17 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING
...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING
...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
```

```
0 BEGIN SL 2 MM
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM
```

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 the simulation 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.

9.5.2 Cycle 20 CONTOUR DATA

ISO programming
G120

Application

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

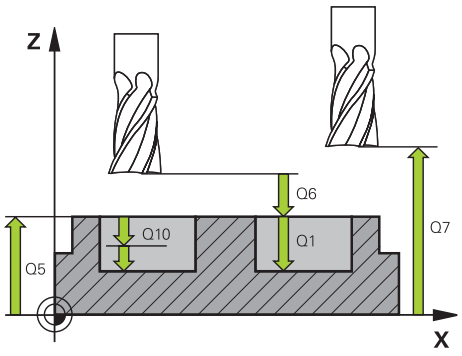
Related topics

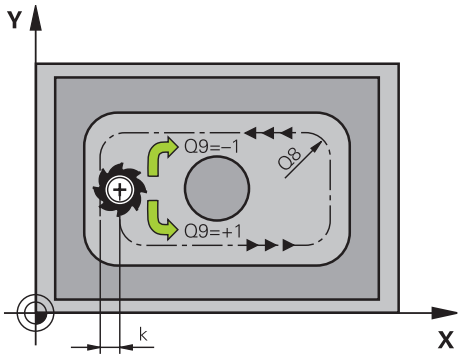
- Cycle **271 OCM CONTOUR DATA** (#167 / #1-02-1)
  Further information: "Cycle 271 OCM CONTOUR DATA (#167 / #1-02-1) ", Page 360

Notes

- This cycle can be executed only 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** x tool radius = 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**

Help graphic	Parameter
	**Q7 Clearance height?** Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). This 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           ;CLIMB MILLING
```

9.5.3 Cycle 21 PILOT DRILLING

ISO programming
G121

Application

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 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
6 The tool then advances with another infeed at the programmed feed rate **F**.
7 The control repeats this procedure (steps 1 to 4) until the total hole depth is reached. The finishing allowance for floor is taken into account
8 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

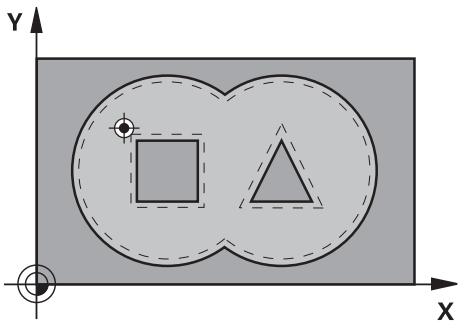
Notes

- This cycle can be executed only 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 the selection option in the action bar. Input: **0...999999.9** or max. **255** characters

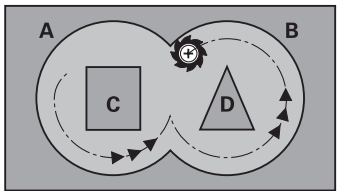
Example

```
11 CYCL DEF 21 PILOT DRILLING ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q13=+0              ;ROUGH-OUT TOOL
```

9.5.4 Cycle 22 ROUGH-OUT

ISO programming
G122

Application



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

Related topics

- Cycle **272 OCM ROUGHING** (#167 / #1-02-1)
  Further information: "Cycle 272 OCM ROUGHING (#167 / #1-02-1)", Page 363

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. There is a danger of collision!

- 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 be executed only 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: Programming and Testing User's Manual

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

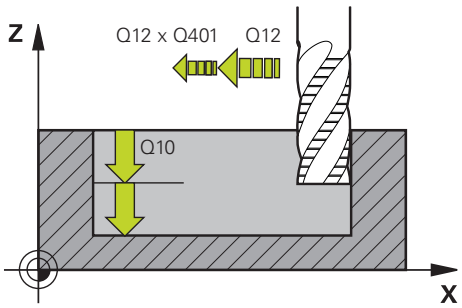
Notes on programming

- 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**) has been 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 control plunges helically using the defined **ANGLE**.
  - If the reciprocation feed is defined in Cycle **22** and no **ANGLE** can be found in the tool table, the control displays an error message.
  - If the geometry conditions do not allow helical plunging (slot geometry), the control tries a reciprocating plunge (the reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**))

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
	**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**
	Q18 or **QS18 Coarse roughing tool?** (optional) Number or name of the tool with which the control has already coarse-roughed the contour. You can use the action bar selection to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself by selecting Name in the action bar. 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
	Q19 Feed rate for reciprocation? (optional) Reciprocation feed rate in mm/min Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? (optional) 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**

Help graphic	Parameter
	Q401 Feed rate factor in %? (optional) 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)? (optional) Define how the control moves the tool during fine roughing: **0**: Between areas that need to be fine-roughed, the control moves the tool along the contour at the current depth. The entry is effective only when the diameter of the fine-roughing tool is larger than or equal to the coarse roughing tool radius. **1**: Between the areas that need to be fine-roughed, the control retracts the tool to the 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             ;CONNECT AREAS DIRECTLY
```

9.5.5 Cycle 23 FLOOR FINISHING

ISO programming
G123

Application

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

Related topics

- Cycle **273 OCM FINISHING FLOOR** (#167 / #1-02-1)
  Further information: "Cycle 273 OCM FINISHING FLOOR (#167 / #1-02-1)", Page 368

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. There is a danger of collision!

- 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

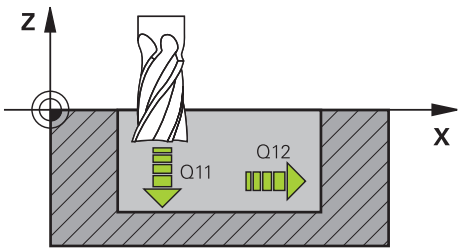
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.
- The approaching radius for pre-positioning to the final depth is permanently defined 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: Programming and Testing User's Manual

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? (optional) 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
```

9.5.6 Cycle 24 SIDE FINISHING

ISO programming
G124

Application

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** if required **ROUGH-OUT**

Related topics

- Cycle **274 OCM FINISHING SIDE** (#167 / #1-02-1)
  Further information: "Cycle 274 OCM FINISHING SIDE (#167 / #1-02-1)", Page 371

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. There is a danger of collision!

- 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 be executed only in the **FUNCTION MODE MILL** machining mode.
- If no finishing allowance was defined in Cycle **20**, the control issues 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: Programming and Testing User's Manual

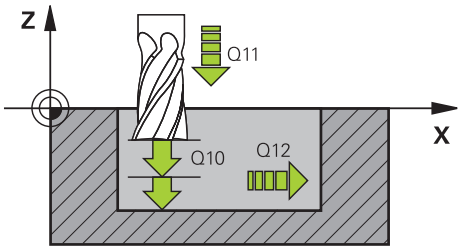
Notes on programming

- 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?** (optional) 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 the action bar. In addition, you can enter the tool name via the Name in the action bar. 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                ;CLIMB MILLING ~
    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
```

9.5.7 Cycle 270 CONTOUR TRAIN DATA

ISO programming
G270

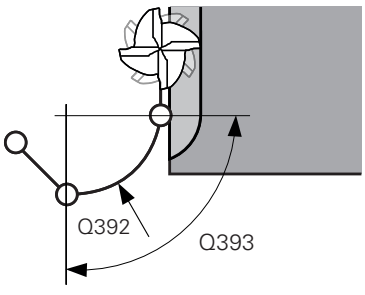
Application

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

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Cycle **270** is DEF-active, which means that it takes effect 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	Parameter
	Q390 Type of approach/departure? Definition of type of approach/departure: **1**: Approach the contour tangentially on a circular arc **2**: Approach the contour tangentially on a straight line **3**: Approach the contour at a right angle **0** and **4**: No approach or departure movement is performed. Input: **1**, **2**, **3**
	Q391 Radius comp. (0=R0/1=RL/2=RR)? Definition of radius compensation: **0**: Machine the defined contour without radius compensation **1**: Machine the defined contour with compensation to the left **2**: Machine the defined contour with compensation to the right Input: **0**, **1**, **2**
	**Q392 App. radius/dep. radius?** Only in effect if a tangential approach on a circular path was selected (**Q390** = 1). Radius of the approach/departure arc Input: **0...99999.9999**
	Q393 Center angle? Only in effect if a tangential approach on a circular path was selected (**Q390** = 1). Angular length of the approach arc Input: **0...99999.9999**
	Q394 Distance from aux. point? Only in effect if a tangential approach on a straight line or a right-angle approach is selected (**Q390** = 2 or **Q390** = 3). Distance to the auxiliary point from which the tool will approach the contour. Input: **0...99999.9999**

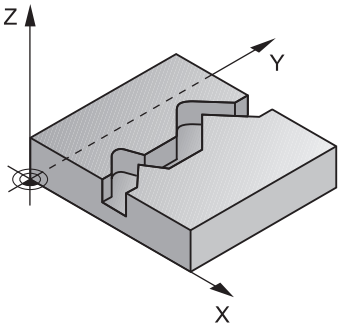
Example

```
11 CYCL DEF 270 CONTOUR TRAIN DATA ~
    Q390=+1                ;TANGENTIAL CIRCLE ~
    Q391=+1                ;RADIUS COMPENSATION, LEFT ~
    Q392=+5                ;RADIUS ~
    Q393=+90               ;CENTER ANGLE ~
    Q394=+0                ;DISTANCE
```

9.5.8 Cycle 25 CONTOUR TRAIN

ISO programming
G125

Application



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. There is a danger of collision!

- 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 be executed only 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: Programming and Testing User's Manual

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
	Q1 Milling depth? Distance between workpiece surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the working plane. 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**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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**
	Q15 Climb or up-cut? up-cut = -1 **+1**: Climb milling **-1**: Up-cut milling **0**: Climb milling and up-cut milling alternately in several infeeds Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q18 or **QS18 Coarse roughing tool?** (optional) Number or name of the tool with which the control has already coarse-roughed the contour. You can use the action bar selection to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself by selecting Name in the action bar. 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 recip-rocating 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? (optional) 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? (optional) 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? (optional) 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 MILLING ~
    Q18=+0                  ;COARSE ROUGHING TOOL ~
    Q446=+0.01              ;RESIDUAL MATERIAL ~
    Q447=+10                ;CONNECTION DISTANCE ~
    Q448=+2                 ;PATH EXTENSION
```

9.5.9 Cycle 275 TROCHOIDAL SLOT

ISO programming
G275

Application

In conjunction with Cycle **14 CONTOUR** , 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.

Enormous amounts of time can also be saved by combining this milling method with the integrated adaptive feed control (**AFC** (#45 / #2-31-1)).

Further information: Programming and Testing User's Manual

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 to the first infeed depth in a reciprocating motion at the plunging angle defined in the tool table. 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 a user-definable infeed (**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 a user-definable infeed (**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. There is a danger of collision!

- 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

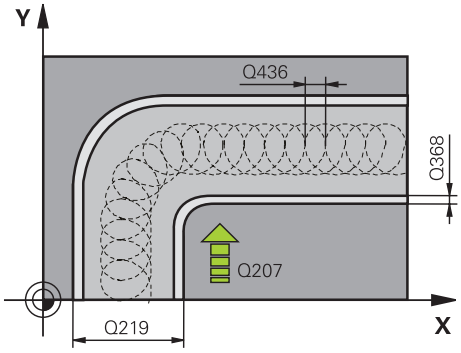
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- In conjunction with Cycle **275**, the control does not require Cycle **20 CONTOUR DATA**.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

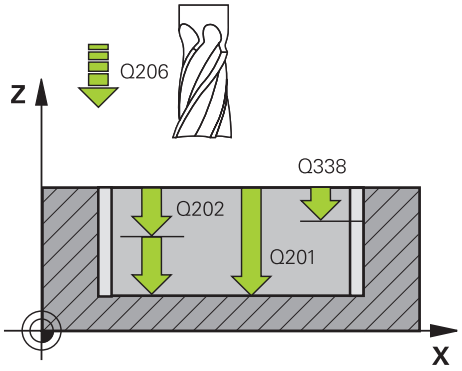
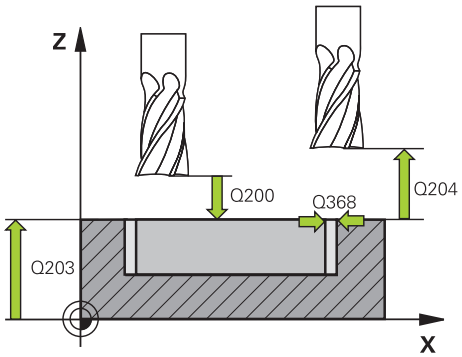
  Further information: Programming and Testing User's Manual

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 executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q219 Width of slot? Enter the width of the slot. This value has an incremental effect. Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. 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. This 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? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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. This 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? (optional) Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q439 Feed rate reference (0-3)? (optional) 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**

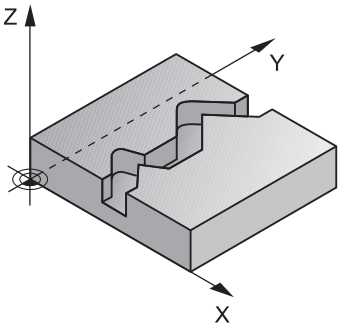
Example

```
11 CYCL DEF 275 TROCHOIDAL SLOT ~
    Q215=+0              ;ROUGHING AND FINISHING ~
    Q219=+10             ;SLOT WIDTH ~
    Q368=+0              ;ALLOWANCE FOR SIDE ~
    Q436=+2              ;INFEED PER REV. ~
    Q207=+500            ;FEED RATE MILLING ~
    Q351=+1              ;CLIMB MILLING ~
    Q201=-20             ;DEPTH ~
    Q202=+5              ;PLUNGING DEPTH ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q338=+0              ;INFEED FOR FINISHING ~
    Q385=+500            ;FINISHING FEED RATE ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q366=+2              ;RECIPROCATING PLUNGE ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q439=+0              ;CENTER POINT PATH
12 CYCL CALL
```

9.5.10 Cycle 276 THREE-D CONT. TRAIN

ISO programming
G276

Application



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 for example 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. There is a danger of collision!

- 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 be executed only 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: Programming and Testing User's Manual

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
	Q1 Milling depth? Distance between workpiece surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the working plane. 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). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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**
	Q15 Climb or up-cut? up-cut = -1 **+1**: Climb milling **-1**: Up-cut milling **0**: Climb milling and up-cut milling alternately in several infeeds Input: **-1**, **0**, **+1**
	Q18 or **QS18 Coarse roughing tool?** (optional) Number or name of the tool with which the control has already coarse-roughed the contour. You can use the action bar selection to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself by selecting Name in the action bar. 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 recip-rocating 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

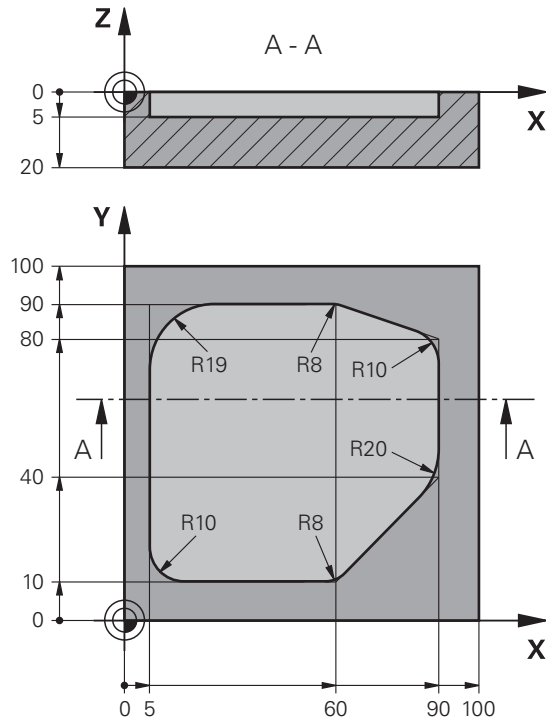
Help graphic	Parameter
	Q446 Accepted residual material? (optional) Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? (optional) 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? (optional) 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 MILLING ~
    Q18=+0                ;COARSE ROUGHING TOOL ~
    Q446=+0.01            ;RESIDUAL MATERIAL ~
    Q447=+10              ;CONNECTION DISTANCE ~
    Q448=+2               ;PATH EXTENSION
```

9.5.11 Programming examples

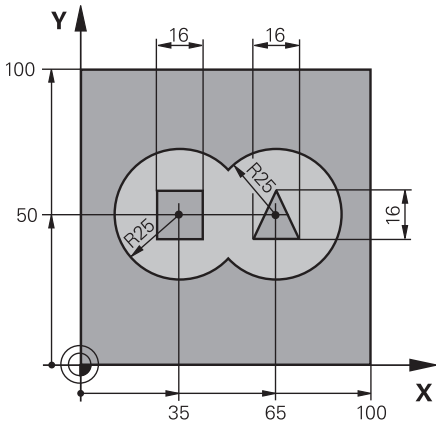
Example: Roughing-out and fine-roughing a pocket with SL Cycles



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 ;CLIMB MILLING	
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 ;TO SETUP CLEARANCE	
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 ;TO SETUP CLEARANCE	
14 CYCL CALL	; Cycle call: fine roughing
15 L Z+200 R0 FMAX	; Retract the tool
16 M30	; End of program run
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	

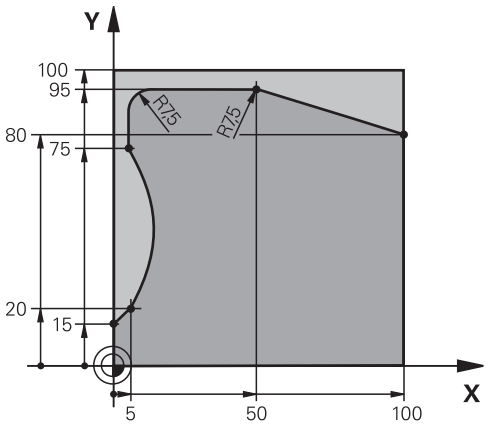
Example: Pilot drilling, roughing and finishing overlapping contours with SL Cycles



0 BEGIN PGM 2 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 "DRILL_D12" Z S2500	; Tool call: drill (diameter: 12)
4 L Z+250 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1 /2 /3 /4	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-20 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0.5 ;ALLOWANCE FOR SIDE ~	
Q4=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+100 ;CLEARANCE HEIGHT ~	
Q8=+0.1 ;ROUNDING RADIUS ~	
Q9=-1 ;UP-CUT MILLING	
8 CYCL DEF 21 PILOT DRILLING ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q13=+0 ;ROUGH-OUT TOOL	
9 CYCL CALL	; Cycle call: pilot drilling
10 L Z+100 R0 FMAX	; Retract the tool
11 TOOL CALL 6 Z S3000	; Tool call: roughing/finishing (D12)
12 CYCL DEF 22 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 ;CONNECT AREAS DIRECTLY	
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 ;CLIMB MILLING ~	
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 run
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 LABEL1	
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 MILLING ~	
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
10 M30	; End of program run
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
```

9.6 Milling contours with OCM cycles (#167 / #1-02-1)

9.6.1 Fundamentals

Application

The OCM cycles include highly efficient roughing or finishing cycles that ease the load on the tool. Using OCM cycles, the control automatically calculates complex movements for milling pockets and islands. Besides pockets and islands, you can also machine open pockets. When roughing, the control will maintain the specified tool angle precisely.

During programming, you can apply the optimal machining parameters from the OCM cutting data calculator directly on the control. The OCM cutting data calculator benefits from an integrated, comprehensive material database. You can adapt the automatically calculated cutting values with regard to the mechanical and thermal load on the tool and transfer them to the roughing cycle.

In order to machine standard shapes, OCM offers various geometric shapes that can then be used as pockets, islands, or boundaries for face milling in conjunction with other OCM cycles.

The OCM cycles are more powerful than Cycles **22** to **24**.

Related topics

- OCM cutting data calculator
  Further information: "OCM cutting data calculator (#167 / #1-02-1)", Page 474
- OCM: Geometric figures
  Further information: "OCM cycles for figure definition", Page 150

Overview of the OCM cycles (#167 / #1-02-1)

Fixed cycles

Cycle		Call	Further information
271	**OCM CONTOUR DATA** ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block	**DEF**-active	Page 360
272	**OCM ROUGHING** ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable	**CALL**-active	Page 363
273	**OCM FINISHING FLOOR** ■ Finishing with finishing allowance for the floor from Cycle **271** ■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances)	**CALL**-active	Page 368

Cycle		Call	Further information
274	**OCM FINISHING SIDE** ■ Finishing with side finishing allowance from Cycle **271**	**CALL**-active	Page 371
277	**OCM CHAMFERING** ■ Deburr the edges ■ Consideration of adjacent contours and walls	**CALL**-active	Page 374

OCM: Geometric figures

Cycle		Call	Further information
1271	**OCM RECTANGLE** ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners	**DEF**-active	Page 153
1272	**OCM CIRCLE** ■ Definition of a circle ■ Input of the circle diameter	**DEF**-active	Page 157
1273	**OCM SLOT / RIDGE** ■ Definition of a groove or ridge ■ Input of the width and the length	**DEF**-active	Page 160
1274	**OCM CIRCULAR SLOT** ■ Definition of a circular slot ■ Input of the width, the pitch circle, and the number of repeats	**DEF**-active	Page 163
1278	**OCM POLYGON** ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners	**DEF**-active	Page 166
1281	**OCM RECTANGLE BOUNDARY** ■ Definition of a bounding rectangle	**DEF**-active	Page 170
1282	**OCM CIRCLE BOUNDARY** ■ Definition of a bounding circle	**DEF**-active	Page 172

Requirements

- Software option Opt. Contour Milling (#167 / #1-02-1)
- Refer to your machine manual. Read and note the functional description of the machine manufacturer. Follow the safety precautions.
- OCM cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always verify the program graphically! This is a simple way to find out whether the program calculated by the control will provide the desired results.

Description of function

Program structure

Program structure: Machining with OCM cycles

The table below shows an example of what a program run with the OCM cycles might look like.

```
0 BEGIN OCM MM
...
12 CONTOUR DEF; Define contour call or figure cycles
...
13 CYCL DEF 271 OCM CONTOUR DATA; Only required for contour definitions
...
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
...
35 CYCL DEF 277 OCM CHAMFERING
36 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
```

Contour definition

OCM figure cycles

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

Further information: "OCM cycles for figure definition", Page 150

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.

Contour formula

Specify the contour with **CONTOUR DEF / SEL CONTOUR** or with the OCM figure 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.

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.

Related topics

- Contour call with a simple contour formula **CONTOUR DEF**
  Further information: "Simple contour formula", Page 107
- Contour call with a complex contour formula **SEL CONTOUR**
  Further information: "Complex contour formula", Page 111
- OCM cycles for figure definition
  Further information: "OCM cycles for figure definition", Page 150

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°.

Positioning logic in OCM cycles

The current tool position is above the clearance height:

1 The control moves the tool to the starting point in the working plane at rapid traverse.
2 The tool moves at **FMAX** to **Q260 CLEARANCE HEIGHT** and then to **Q200 SET-UP CLEARANCE**
3 The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**.

The current tool position is below the clearance height:

1 The control moves the tool to **Q260 CLEARANCE HEIGHT** at rapid traverse.
2 At **FMAX**, the tool moves to the starting point in the working plane and then to **Q200 SET-UP CLEARANCE**
3 The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**

Programming and operating notes:

- **Q260** The control uses the **CLEARANCE HEIGHT** from Cycle **271 OCM CONTOUR DATA** or from the figure cycles.
- **Q260 CLEARANCE HEIGHT** is effective only when the position of the safe height is above the safety distance.

Removing residual material

When roughing, these cycles allow you to use larger tools for the first roughing passes and then smaller tools to remove the residual material. During finishing the control will take into account the material roughed out, thus preventing the finishing tool from being overloaded.

Further information: "Example: Open pocket and fine roughing with OCM cycles", Page 378

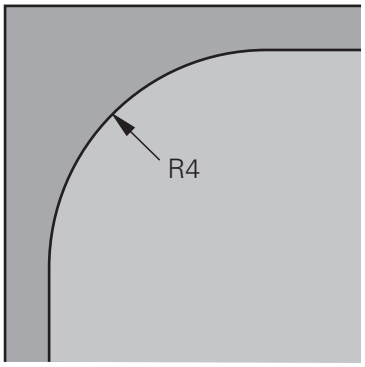
- If residual material remains in the inside corners after roughing, then use a smaller rough-out tool or define an additional roughing operation with a smaller tool.
- If the inside corners cannot be roughed out completely, the control may damage the contour during chamfering. In order to prevent damage to the contour, follow the procedure described below.

Procedure regarding residual material in inside corners

The example describes the inside machining of a contour by using several tools with radii greater than the programmed contour. Although the radius of the tools used becomes smaller, residual material remains in the inside corners after roughing. The control takes this residual material into account during the subsequent finishing and chamfering operations.

In the example, you use the following tools:

- **MILL_D20_ROUGH**, Ø 20 mm
- **MILL_D10_ROUGH**, Ø 10 mm
- **MILL_D6_FINISH**, Ø 6 mm
- **NC_DEBURRING_D6**, Ø 6 mm



Inside corner with a radius of 4 mm in this example

Roughing

- Rough the contour with the tool **MILL_D20_ROUGH**
- The control considers the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 12 mm during initial roughing.

TOOL CALL Z 10	; "MILL_D20_ROUGH"
CYCL DEF 271 OCM CONTOUR DATA	
... **;**	
Q578 = 0.2 **;INSIDE CORNER FACTOR**	; Resulting inner radius = R_T+ (**Q578** * R_T) = 10 + (0.2 *10) = 12
... **;**	
CYCL DEF 272 OCM ROUGHING	
... **;**	
...	

- Then rough the contour with the smaller tool **MILL_D10_ROUGH**
- The control takes into account the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 6 mm during initial roughing.

TOOL CALL Z 5	; "MILL_D10_ROUGH"
CYCL DEF 271 OCM CONTOUR DATA	
... **;**	
Q578 = 0.2 **;INSIDE CORNER FACTOR**	; Resulting inner radius = R_T+ (**Q578** * R_T) = 5 + (0.2 *5) = 6
... **;**	
CYCL DEF 272 OCM ROUGHING	
... **;**	
Q438=-1 **;ROUGH-OUT TOOL**	; The tool used last is assumed to be the rough-out tool
... **;**	
...	

Finishing

- Finish the contour with the tool **MILL_D6_FINISH**

> This finishing tool would allow inside radii of 3.6 mm. This means that the finishing tool would be capable of machining the defined inside radii of 4 mm. However, the control takes into account the residual material of the rough-out tool **MILL_D10_ROUGH**. The control machines the contour with the previous roughing tool's inside radii of 6 mm. Thus, the finishing cutter will be protected from overload.

TOOL CALL Z 23	; "MILL_6_FINISH"
CYCL DEF 271 OCM CONTOUR DATA	
... **;**	
Q578 = 0.2 **;INSIDE CORNER FACTOR**	; Resulting inner radius = R_T+ (**Q578** * R_T) = 3 + (0.2 *3) = 3.6
... **;**	
CYCL DEF 274 OCM FINISHING SIDE	
... **;**	
Q438=-1 **;ROUGH-OUT TOOL**	; The tool used last is assumed to be the rough-out tool
... **;**	
...	

- Chamfering the contour: When defining the cycle, you must define the last rough-out tool of the roughing operation.

> If you use the finishing tool as a roughing tool, the control will damage the contour. In this case, the control assumes that the finishing cutter machined the contour with inside radii of 3.6 mm. However, the finishing cutter has limited the inside radii to 6 mm based on the previous roughing operation.

TOOL CALL Z 170	; "NC_DEBURRING_D6"
CYCL DEF 277 OCM CHAMFERING	
... **;**	
QS438 = 6 **;ROUGH-OUT TOOL**	; Rough-out tool "MILL_D10_ROUGH" of the last roughing operation
... **;**	
...	

9.6.2 Cycle 271 OCM CONTOUR DATA (#167 / #1-02-1)

ISO programming
G271

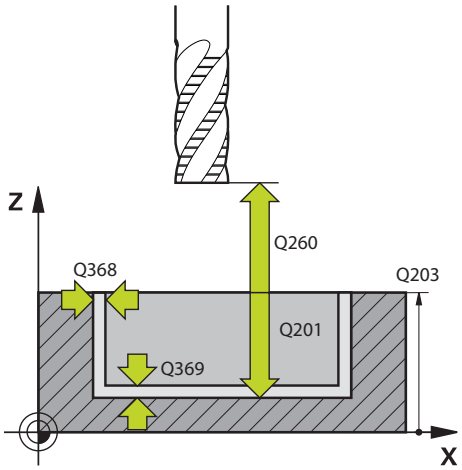
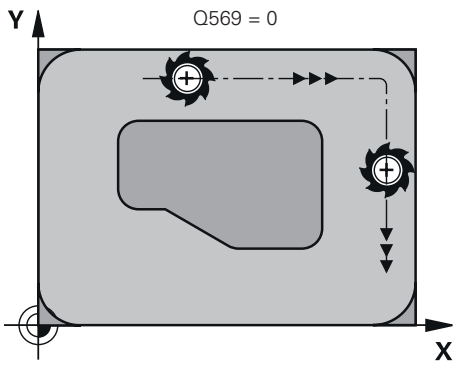
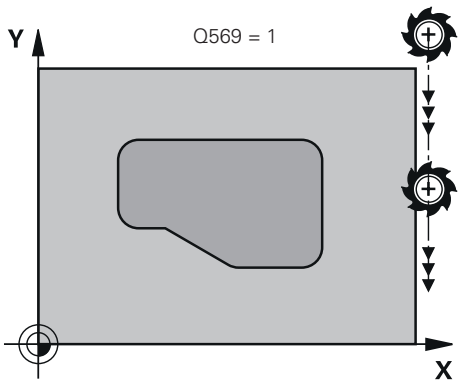
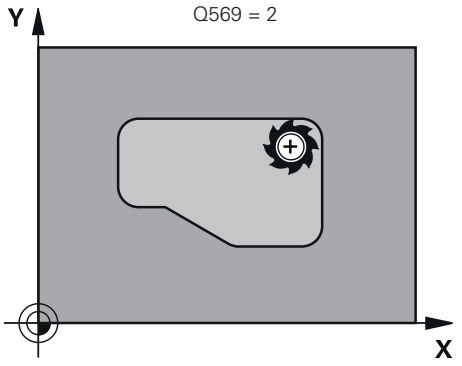
Application

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 be executed only 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. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** (or **PREDEF**)
	Q578 Radius factor on inside corners? The tool radius multiplied by **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. 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                ;POCKET
```

9.6.3 Cycle 272 OCM ROUGHING (#167 / #1-02-1)

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 (#167 / #1-02-1)", Page 474

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

Further information: "Positioning logic in OCM cycles", Page 356

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 364
- **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**)
8 Finally, the tool moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 CLEARANCE HEIGHT**

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 INFEED STRATEGY**
- **ANGLE**
- **RCUTS**
- $\mathbf{R_{corr}}$ (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

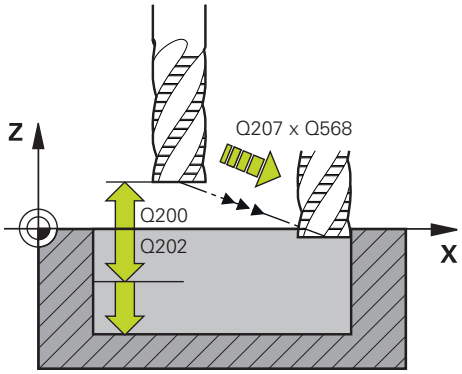
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for floor 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.

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

Notes on programming

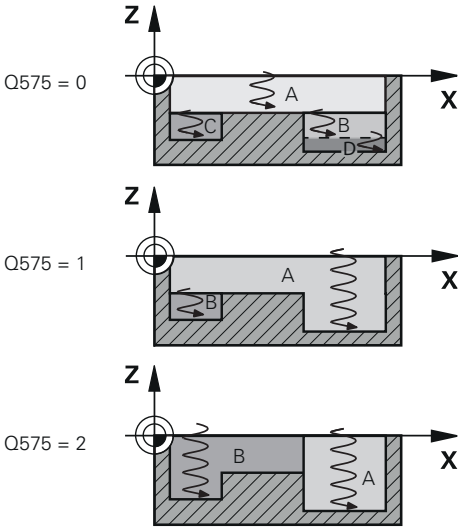
- **CONTOUR DEF** / **SEL CONTOUR** will reset the tool radius that was used last. If you run this machining cycle with **Q438** = –1 after **CONTOUR DEF** / **SEL CONTOUR**, the control assumes that no pre-machining has taken place yet.
- If the path overlap factor **Q370 < 1**, a value of less than 1 is also recommended for the plunging factor **Q579** .
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

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**)

Help graphic	Parameter
	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 the action bar. In addition, you can enter the tool name via the Name in the action bar. 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
	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? (optional) 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? (optional) 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)? (optional)

Type of downfeed:

0: The cycle machines the contours step by step from the top down, level by level. The control starts with the highest contour or plunging depth and mills all pockets down to this depth. After that, it moves to the next lower level and machines only the remaining pockets which have not yet reached their final depth. This procedure is repeated until all pockets have reached their respective final depth.

1: The control machines the contour levels from bottom to top. The control does not always start with the lowest contour level. 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 levels from bottom to top. The control does not always start with the lowest contour level. The machining sequence is calculated so that the length of the cutting edge is used to the maximum. The resulting total plunging path is thus often greater than with strategy **1**. Depending on **Q568**, a shorter machining time may result.

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 MILLING ~
    Q576=+0          ;SPINDLE SPEED ~
    Q579=+1          ;PLUNGING FACTOR S ~
    Q575=+0          ;FROM TOP TO BOTTOM
```

9.6.4 Cycle 273 OCM FINISHING FLOOR (#167 / #1-02-1)

ISO programming
G273

Application

With Cycle **273 OCM FINISHING FLOOR**, you can clear 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** or Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable

Cycle sequence

1 The tool uses positioning logic to move to the starting point
Further information: "Positioning logic in OCM cycles", Page 356
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 moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 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!

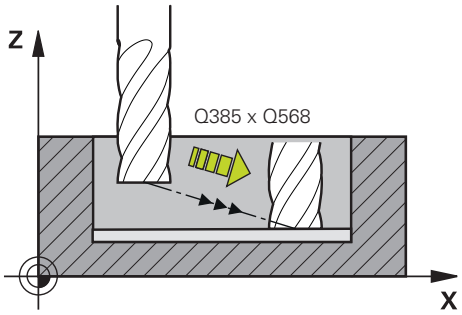
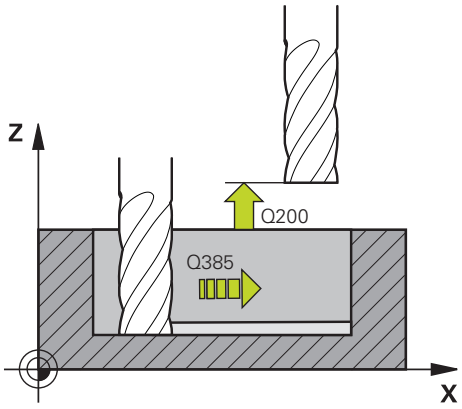
- ▶ Run a simulation to verify the machining sequence and the contour
- ▶ Use tools without a corner radius **R2** where possible

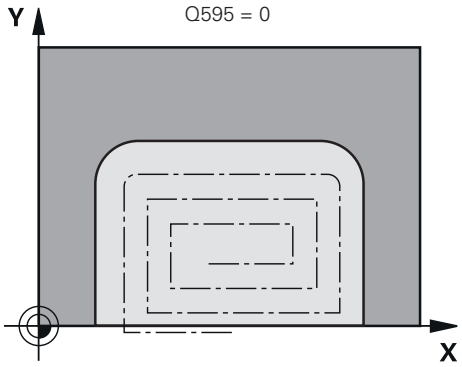
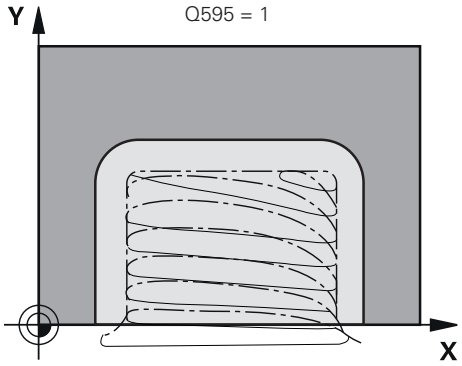
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- You can execute this cycle with a grinding tool. The permitted grinding tools include the grinding pin **(PIN)** and the cup wheel **(CUP)**.
  The tooth length **LCUTS** is determined as follows:
  - Grinding pin: **LCUTS** = width **B**
  - Cup wheel: **LCUTS** = Depth **G * sin(GAMMA)**
- The control automatically calculates the starting point for floor 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 the action bar. In addition, you can enter the tool name via the Name in the action bar. 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)?** (optional) Machining strategy for finishing **0**: Equidistant strategy = constant distances between paths **1**: Strategy with constant contact angle Input: **0**, **1**
	Q577 Factor for appr./dept. radius? (optional) 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                 ;CONSTANT TOOL CONTACT ANGLE ~
    Q577=+0.2               ;APPROACH RADIUS FACTOR
```

9.6.5 Cycle 274 OCM FINISHING SIDE (#167 / #1-02-1)

ISO programming
G274

Application

With Cycle **274 OCM FINISHING SIDE**, you can clear 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** or **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable

Cycle sequence

1 The tool uses positioning logic to move to the starting point
2 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
Further information: "Positioning logic in OCM cycles", Page 356
3 The control then moves the tool to the first plunging depth using the feed rate for plunging
4 The tool approaches and moves along the contour helically on a tangential arc until the entire contour is finished. Each subcontour is finished separately
5 Finally, the tool moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 CLEARANCE HEIGHT**

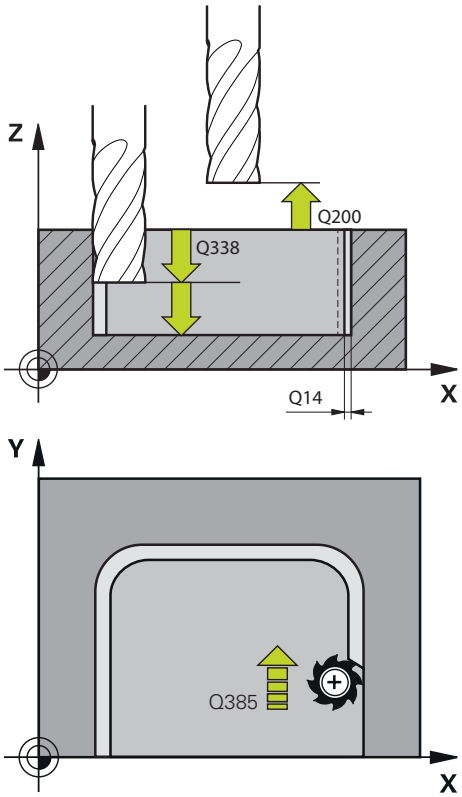
Notes

- This cycle can be executed only 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.
Further information: Programming and Testing User's Manual

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?** Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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 the action bar. In addition, you can enter the tool name via the Name in the action bar. 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                ;INFEED FOR FINISHING ~
    Q385=+500              ;FINISHING FEED RATE ~
    Q253=+750              ;F PRE-POSITIONING ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q14=+0                 ;ALLOWANCE FOR SIDE ~
    Q438=-1                ;ROUGH-OUT TOOL ~
    Q351=+1                ;CLIMB MILLING
```

9.6.6 Cycle 277 OCM CHAMFERING (#167 / #1-02-1)

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.

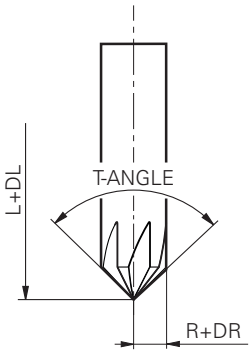
Requirements

Before calling Cycle **277**, you need to program the following cycles:

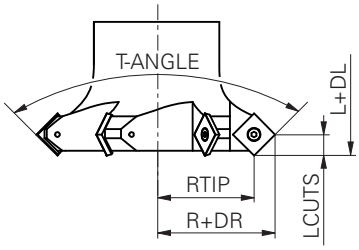
- **CONTOUR DEF / SEL CONTOUR** or **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA** or the standard geometric elements **12xx**
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable

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

Chamfer cutter	Chamfer cutter with indexable insert



- **L** + **DL**: Overall length up to the theoretical tip
- **R** + **DR**: Definition of the overall tool radius
- **T-ANGLE** : Point angle of the tool
- **LCUTS**: Tooth length in the tool axis(optional)



- **L** + **DL**: Overall length
- **R** + **DR**: Definition of the overall tool radius
- **T-ANGLE** : Point angle of the tool
- **R-TIP**: Radius at the tip
- **LCUTS**: Tooth length in the tool axis(optional)

Cycle sequence

1 The tool uses positioning logic to move to the starting point. This point is determined automatically based on the programmed contour.
2 In the next step, the tool moves at **FMAX** to set-up clearance **Q200.**
3 Then, the tool plunges vertically to **Q353 DEPTH OF TOOL TIP**.
4 The tool approaches the contour in a tangential or vertical movement (depending on the available space).
5 Depending on the definition in **Q240 NUMBER OF CUTS**, the tool approaches the first stepover or the entire chamfer width.
6 For machining the chamfer, the tool uses the milling feed rate **Q207**.
7 Then, the tool is retracted from the contour in a tangential or vertical movement (depending on the available space).
8 If there are several contours, all of them will be machined. The tool is positioned at clearance height after each contour and then moves to the next starting point.
9 Depending on the definition in **Q240**, the tool approaches the workpiece laterally; steps 5 to 8 are repeated until the entire programmed contour has been chamfered.
10 Then, the tool moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 CLEARANCE HEIGHT**.

Further information: "Positioning logic in OCM cycles", Page 356

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for chamfering. The starting point depends on the available space.
- The control monitors the tool radius. Adjacent walls machined with Cycle **271 OCM CONTOUR DATA** or with the **12xx** figure cycles will remain intact.
- The cycle monitors for damage to the contour floor from the tool tip. This tool tip results from the radius **R**, the radius of the tool tip **R_TIP**, and the point angle **T-ANGLE**.
- Keep in mind that the active tool radius of the chamfering tool must be smaller than 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. This tool radius results from **T-ANGLE** and **R_TIP** from the tool table.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

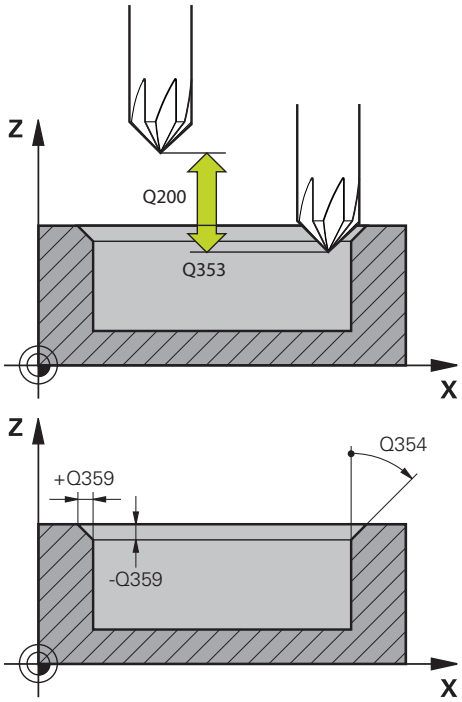
  Further information: Programming and Testing User's Manual
- If the roughing operations have not completely removed the material before chamfering, you need to define the last roughing tool in **QS438 ROUGH-OUT TOOL**, in order to prevent damage to the contour.

  Further information: "Procedure regarding residual material in inside corners", Page 357

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 the action bar. In addition, you can enter the tool name via the Name in the action bar. 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**
	Q240 Number of cuts? (optional) Number of infeeds until the chamfer size is attained The control retains the same depth for all infeeds and shifts the tool only laterally. The control divides the cuts in such a way that a constant chip cross section results over all infeeds. **1:** Machining in one infeed **2-99**: Machining in several infeeds Input: **1...99**

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 MILLING ~
    Q354=+0                 ;CHAMFER ANGLE ~
    Q240=+1                 ;NUMBER OF CUTS
```

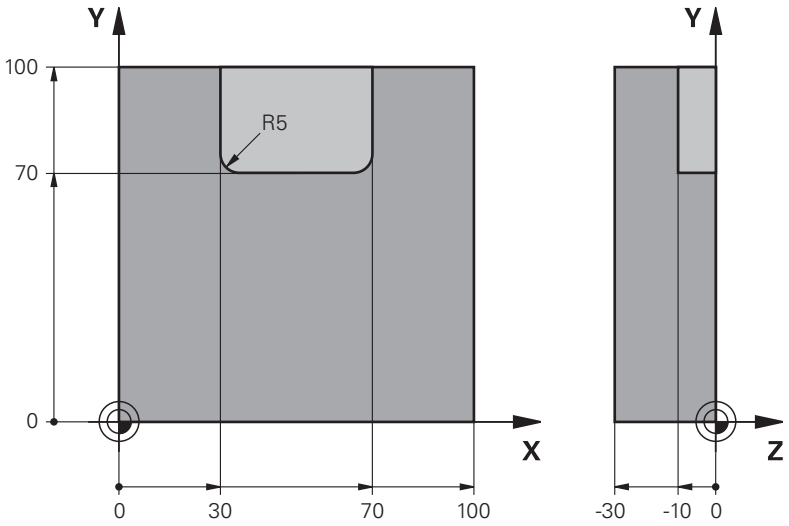
9.6.7 Programming examples

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 MILLING ~	
Q576=+6500 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;FROM TOP TO BOTTOM	
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 MILLING ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;FROM TOP TO BOTTOM	
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 ;CONSTANT TOOL CONTACT ANGLE ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
16 CYCL CALL	; Cycle call
17 CYCL DEF 274 OCM FINISHING SIDE ~	

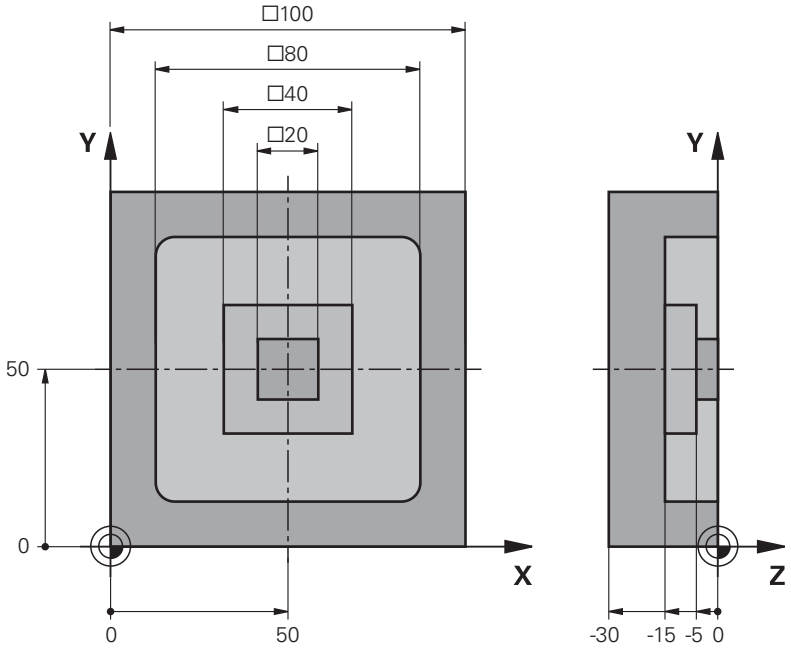
Q338=+0 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB MILLING	
18 CYCL CALL	; Cycle call
19 M30	; End of program run
20 LBL 1	; Contour subprogram 1
21 L X+0 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 ;POCKET	
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 MILLING ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;FROM BOTTOM TO TOP	
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 ;CONSTANT TOOL CONTACT ANGLE ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 CYCL CALL	; Cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+5 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB MILLING	
14 CYCL CALL	; Cycle call
15 M30	; End of program run
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	

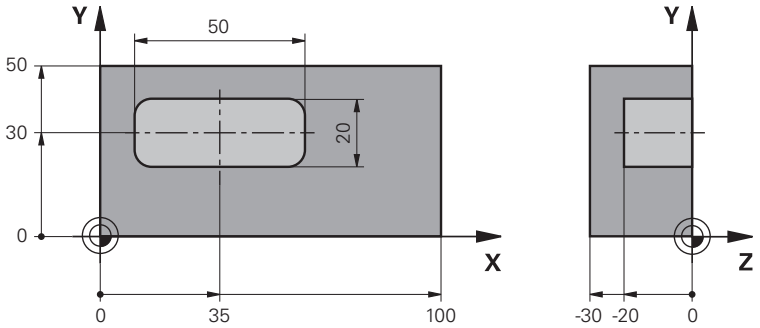
23 LBL 2	; Contour subprogram 2
24 L X-10 Y-10	
25 L X+10	
26 L Y+10	
27 L X-10	
28 L Y-10	
29 LBL 0	
30 LBL 3	; Contour subprogram 3
31 L X-20 Y-20	
32 L X+20	
33 L Y+20	
34 L X-20	
35 L Y-20	
36 LBL 0	
37 END PGM OCM_DEPTH MM	

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=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB MILLING ~	
Q576=+8000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;FROM BOTTOM TO TOP	
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 ;FROM BOTTOM TO TOP	
12 L X+0 Y+0 R0 FMAX M99	; Cycle call
13 M30	; End of program run
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	

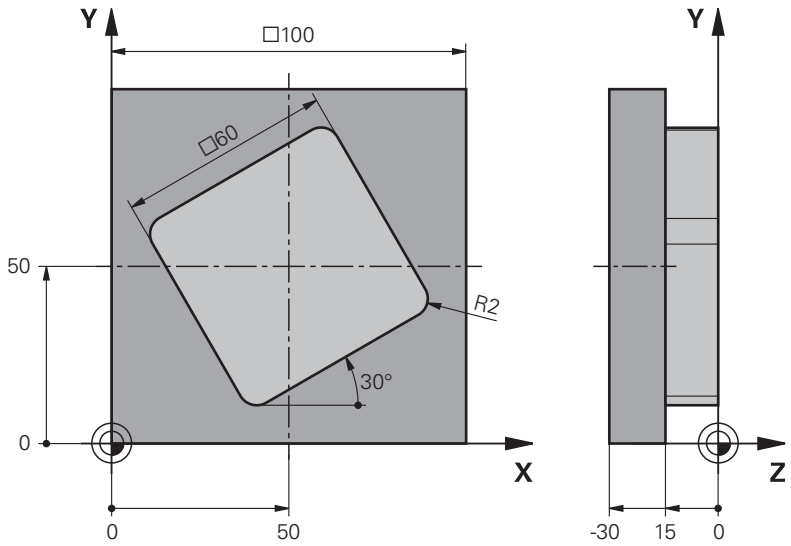
20 LBL 0	
21 LBL 2	; Contour subprogram 2
22 L X+10 Y+30	
23 L Y+40	
24 RND R5	
25 L X+60	
26 RND R5	
27 L Y+20	
28 RND R5	
29 L X+10	
30 RND R5	
31 L Y+30	
32 LBL 0	
33 END PGM FACE_MILL MM	

Example: Contour with OCM figure cycles

The following NC program illustrates the use of OCM cycles. Machining includes roughing and finishing of an 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 ;ISLAND ~	
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 ;INCREMENTAL TO ISLAND ~	
Q655=+0 ;SHIFT 1 ~	
Q656=+0 ;SHIFT 2	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.4 ;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 MILLING ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;FROM BOTTOM TO TOP	
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 ~	

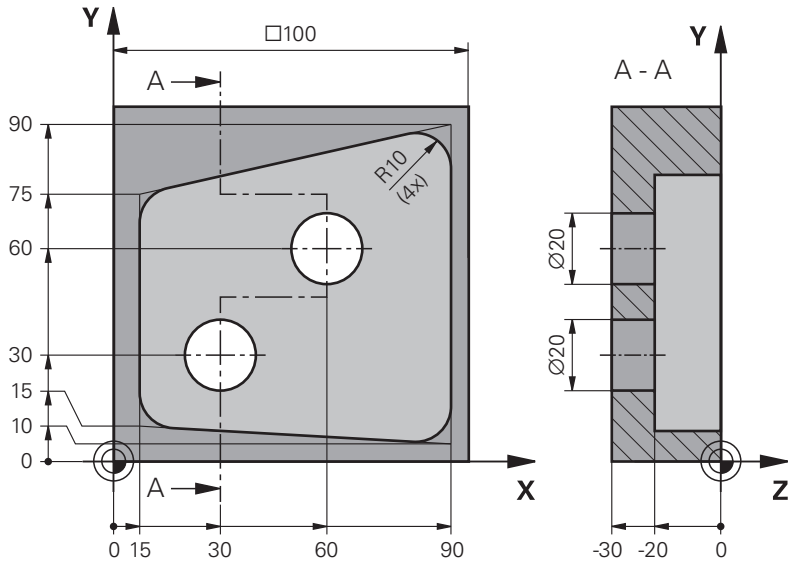
Q438=+4 ;ROUGH-OUT TOOL ~	
Q595=+1 ;CONSTANT TOOL CONTACT ANGLE ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+15 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB MILLING	
14 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
15 M30	; End of program run
16 END PGM OCM_FIGURE MM	

Example: void areas with OCM cycles

The following NC program shows how to define void areas by using OCM cycles. Two circles from the previous machining operation are used to define void areas in **CONTOUR DEF**. The tool plunges perpendicularly within the void area.

Program sequence

- Tool call: drill (diameter: 20 mm)
- Define Cycle **200**
- Tool call: roughing cutter (diameter: 14 mm)
- Define **CONTOUR DEF** with void areas
- Define Cycle **271**
- Define and call Cycle **272**



0 BEGIN PGM VOID_1 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 206 Z S8000 F900	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-30 ;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=+1 ;CUTTING EDGE	
6 L X+30 Y+30 R0 FMAX M99	
7 L X+60 Y+60 R0 FMAX M99	
8 TOOL CALL 7 Z S7000 F2000	; Tool call (diameter: 14 mm)
9 L Z+100 R0 FMAX M3	
10 CONTOUR DEF P1 = LBL 1 V1 = LBL 2 V2 = LBL 3	; Definition of contour and void area
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 ;POCKET	
12 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.441 ;TOOL PATH OVERLAP ~	
Q207=+6000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB MILLING ~	
Q576=+13626 ;SPINDLE SPEED ~	
Q579=+1 ;PLUNGING FACTOR S ~	
Q575=+2 ;MAXIMUM PLUNGING DEPTH	
13 CYCL CALL	
14 M30	; End of program run
15 LBL 1	; Contour subprogram 1
16 L X+90 Y+50	

17 L Y+10	
18 RND R10	
19 L X+10 Y+15	
20 RND R10	
21 L Y+75	
22 RND R10	
23 L X+90 Y+90	
24 RND R10	
25 L Y+50	
26 LBL 0	
27 LBL 2	; Void area 1
28 CC X+30 Y+30	
29 L X+40 Y+30	
30 C X+40 Y+30 DR-	
31 LBL 0	
32 LBL 3	; Void area 2
33 CC X+60 Y+60	
34 L X+70 Y+60	
35 C X+70 Y+60 DR-	
36 LBL 0	
37 END PGM VOID_1 MM	

9.7 Milling planes

9.7.1 Cycle 232 FACE MILLING

ISO programming
G232

Application

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

Related topics

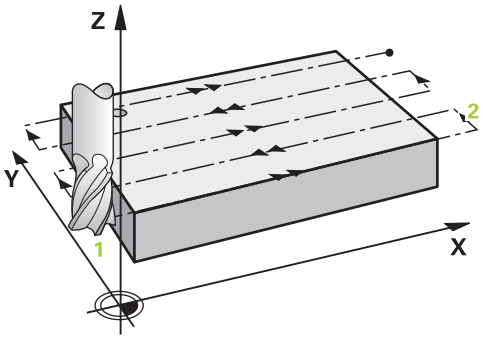
- Cycle **233 FACE MILLING**
  Further information: "Cycle 233 FACE MILLING ", Page 397

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**.

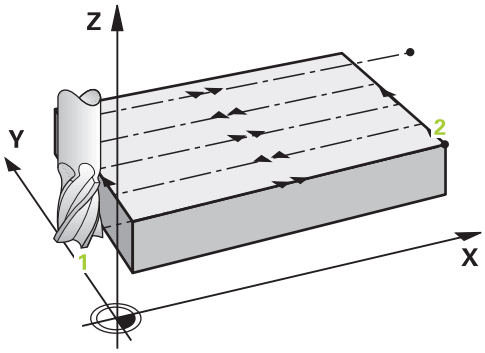
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

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.

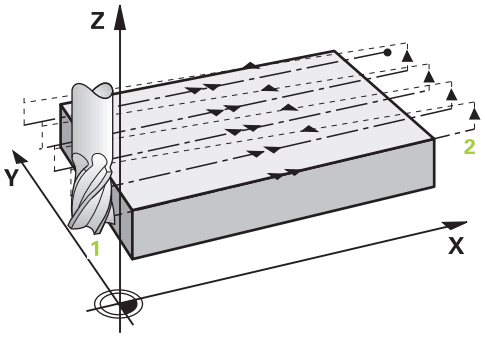
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

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

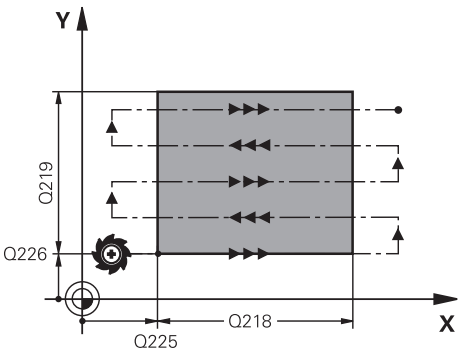
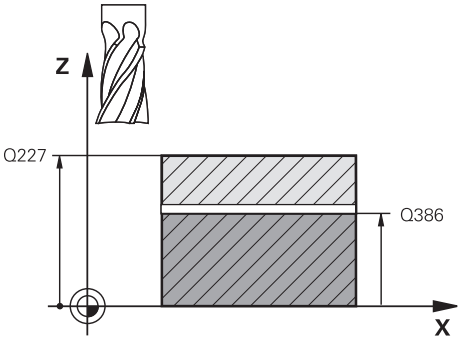
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.

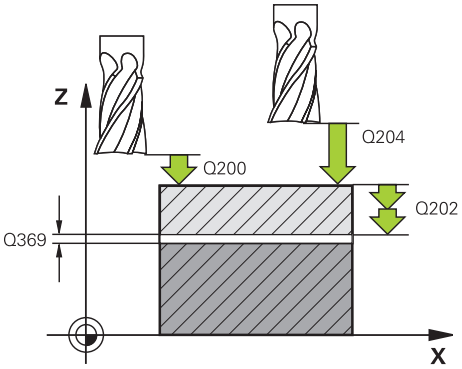
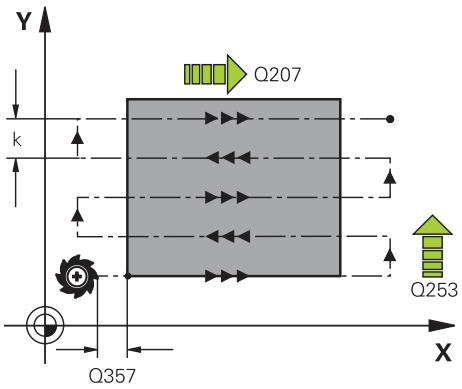
Notes on programming

- 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. This 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. This 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. This 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. This 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? Finishing allowance in depth which remains after roughing. 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
```

9.7.2 Cycle 233 FACE MILLING

ISO programming
G233

Application

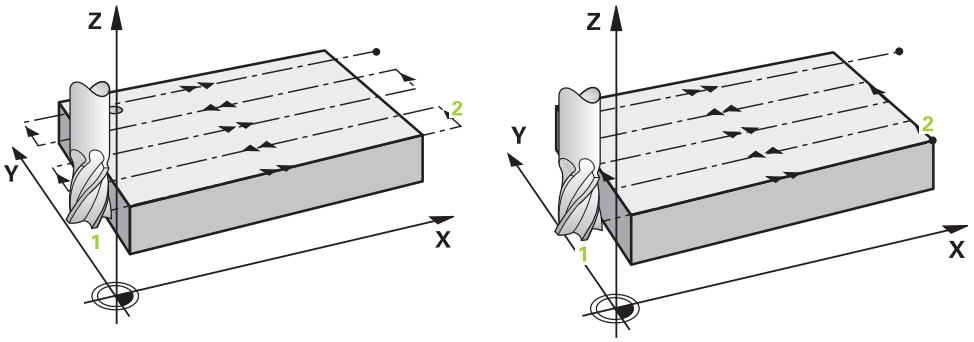
With Cycle **233**, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

- **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

Related topics

- Cycle **232 FACE MILLING**
  Further information: "Cycle 232 FACE MILLING ", Page 390

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.

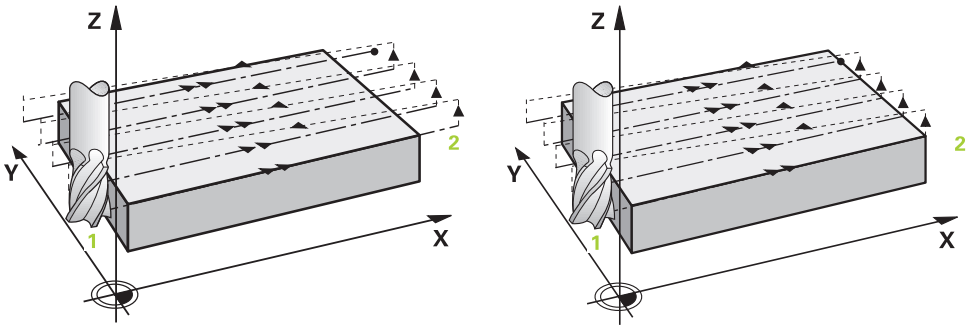
8 The control then positions the tool at rapid traverse **FMAX** back to starting point **1**.

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 **Q207** 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 **Q370** and the set-up clearance to the side **Q357**.

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**.

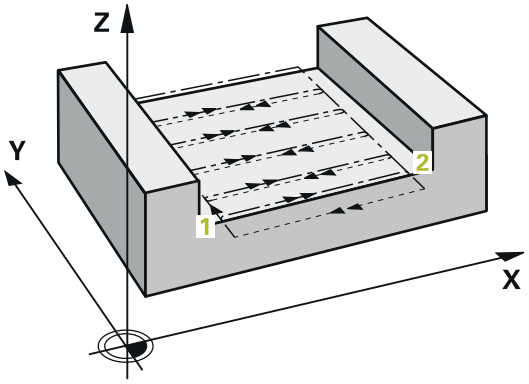
8 If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.

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—with lateral limitation

If you program a lateral limitation, the control might not be able to perform movements outside of the contour. In this case the cycle runs 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 **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 moves the tool to the approach position of the next path at **Q253 F PRE-POSITIONING**.

6 Steps 3 to 5 are repeated until the entire surface is milled.

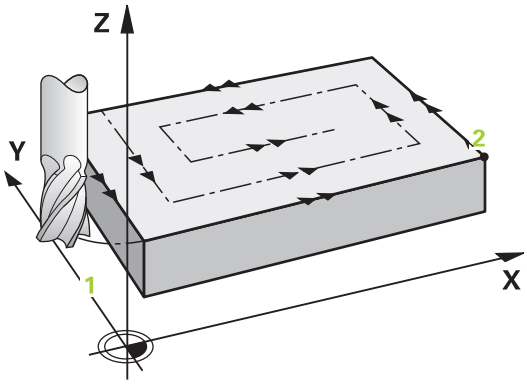
7 If more than one infeed depth is programmed, the control moves the tool at the end of the last path to the set-up clearance **Q200** and positions in the working plane to the next approach position.

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.

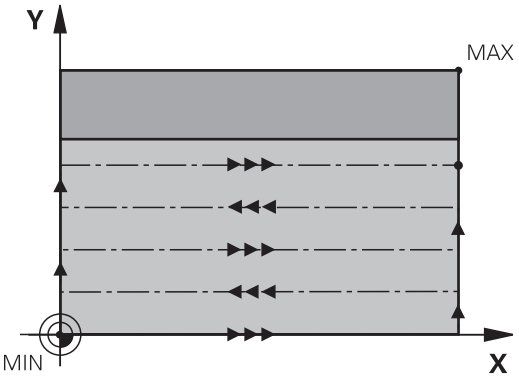
6 The process is repeated until the programmed surface has been completed. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**.

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

Risk of collision!

In Cycle **Q227**, if you enter **STARTNG PNT 3RD AXIS** deeper than **Q386 END POINT 3RD AXIS**, the control will reverse the calculation of pre-positioning. The tool moves at rapid traverse in the tool axis to a set-up clearance **below** the workpiece surface! Risk of collision!

- **Q386** Enter **END POINT 3RD AXIS** deeper than **Q227 STARTNG PNT 3RD AXIS**
- Use 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 be executed only 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.

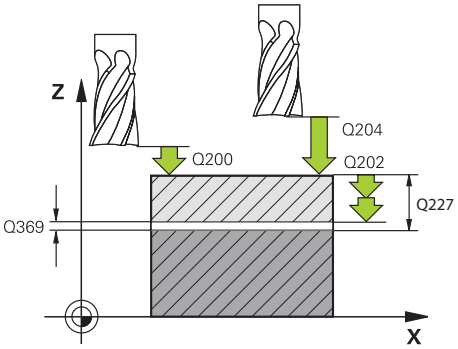
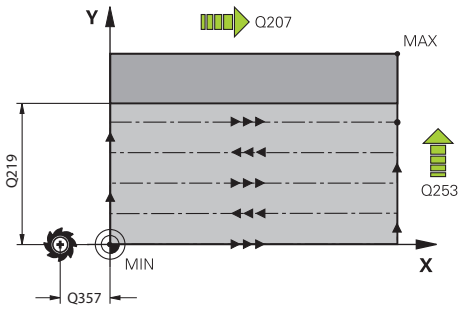
Notes on programming

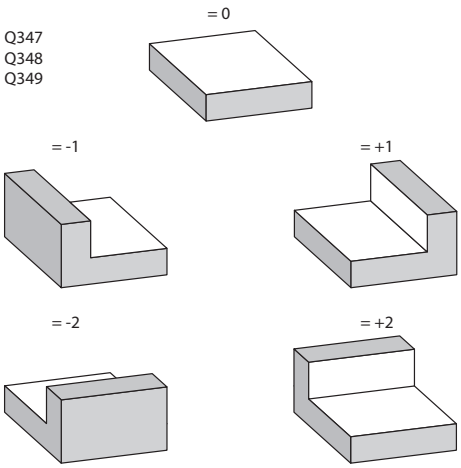
- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- If 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
	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 executed only if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q389 Machining strategy (0-4)? Specify how the control machines 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**: Machining line by line, retraction and stepover at positioning feed rate outside the surface to be machined **3**: Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined **4**: Helical machining, uniform infeed from the outside toward the inside Input: **0**, **1**, **2**, **3**, **4**
	Q350 Milling direction? Axis in the working plane that defines the machining direction: **1**: Main axis = Machining direction **2**: Secondary axis = Machining direction Input: **1**, **2**

Help graphic	Parameter
	**Q218 First side length?** 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: **-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**
	Q227 Starting point in 3rd axis? Coordinate of the workpiece surface used to calculate the infeeds. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. 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**
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q367 Surface position (-1/0/1/2/3/4)? (optional) Position of the surface relative to the position of the tool when the cycle is called: **-1**: Tool position = Current position **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: **-1**, **0**, **+1**, **+2**, **+3**, **+4**

Example

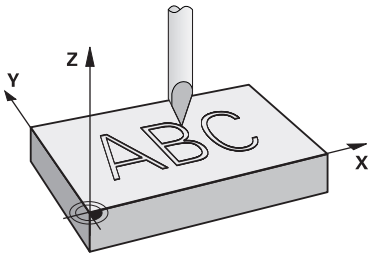
```
11 CYCL DEF 233 FACE MILLING ~
    Q215=+0          ;ROUGHING AND FINISHING ~
    Q389=+2          ;LINE-BY-LINE WITH OVERRUN ~
    Q350=+1          ;MAIN AXIS ~
    Q218=+60         ;FIRST SIDE LENGTH ~
    Q219=+20         ;2ND SIDE LENGTH ~
    Q227=+0          ;STARTNG PNT 3RD AXIS ~
    Q386=+0          ;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          ;NO LIMITATION ~
    Q348=+0          ;NO LIMITATION ~
    Q349=+0          ;NO LIMITATION ~
    Q220=+0          ;CORNER RADIUS ~
    Q368=+0          ;ALLOWANCE FOR SIDE ~
    Q338=+0          ;INFEED FOR FINISHING ~
    Q367=-1          ;SURFACE POSITION
12 L  X+50  Y+50 R0 FMAX M99
```

9.8 Engraving

9.8.1 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**.

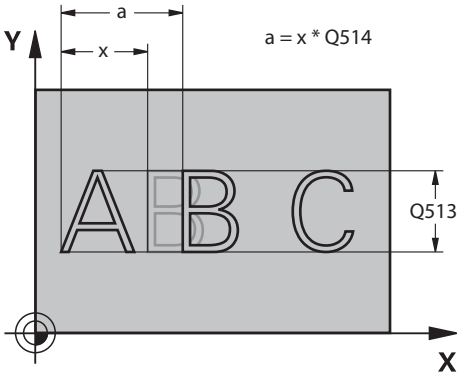
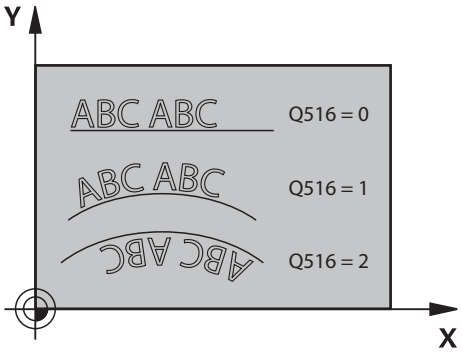
Notes

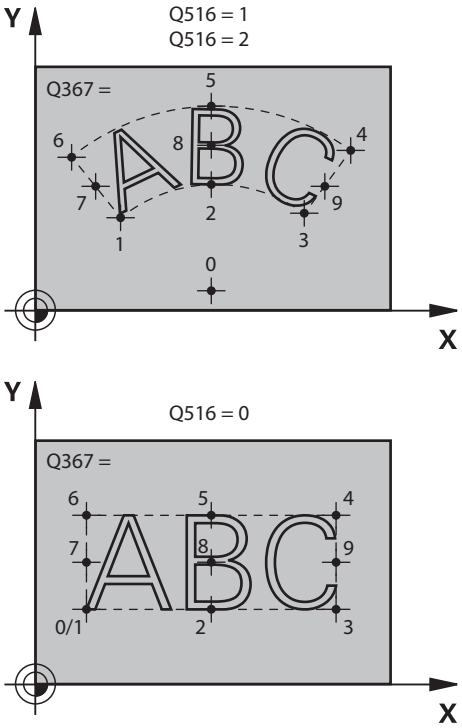
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.

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
	QS500 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
	**Q513 Character height?** Height of the characters to be engraved in mm Input: **0...999.999**
	Q514 Character spacing factor? The width of the characters varies. **X** = width of the character + default spacing. This factor allows you to influence the spacing. **Q514=0/1**: Default spacing between the characters **Q514>1**: The spacing between the characters is expanded. **Q514<1**: The spacing between the characters is reduced. This can lead to overlapping characters. Input: **0...10**
	Q515 Font? **0**: Font **DeJaVuSans** **1**: Font **LiberationSans-Regular** Input: **0**, **1**
	**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. This 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)?** (optional) 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? (optional) 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? (optional) 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 ~
    QS500=""                ;ENGRAVING TEXT ~
    Q513=+10                ;CHARACTER HEIGHT ~
    Q514=+0                 ;SPACE FACTOR ~
    Q515=+0                 ;DEJAVUSANS ~
    Q516=+0                 ;STRAIGHT ALIGNMENT ~
    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	Character
%ae	ä
%oe	ö
%ue	ü
%AE	Ä
%OE	Ö
%UE	Ü
%ss	ß
%D	ø
%at	@
%CE	CE
%Euro	€
%deg	°
%Copyright	©

Non-printable characters

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. To 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	Format
%time00	DD.MM.YYYY hh:mm:ss
%time01	D.MM.YYYY h:mm:ss
%time02	D.MM.YYYY h:mm
%time03	D.MM.YY h:mm
%time04	YYYY-MM-DD hh:mm:ss
%time05	YYYY-MM-DD hh:mm
%time06	YYYY-MM-DD h:mm
%time07	YY-MM-DD h:mm
%time08	DD.MM.YYYY
%time09	D.MM.YYYY
%time10	D.MM.YY
%time11	YYYY-MM-DD
%time12	YY-MM-DD
%time13	hh:mm:ss
%time14	h:mm:ss
%time15	h:mm
%time99	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
%main0	Full path of the active NC program	**TNC:\MILL.h**
%main1	Path to the directory of the active NC program	**TNC:**
%main2	Name of the active NC program	**MILL**
%main3	File type of the active NC program	**.H**
%prog0	Full path of the called NC program	**TNC:\HOUSE.h**
%prog1	Path to the directory of the called NC program	**TNC:**
%prog2	Name of the called NC program	**HOUSE**
%prog3	File type of the active NC program	**.H**

Engraving the counter reading

You can use Cycle **225** to engrave the current counter reading as found on the PGM tab of the **Status** workspace.

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: Programming and Testing User's Manual

Operating notes

- In the simulation, the control only simulates the counter reading you specified directly in the NC program. The counter reading from the program run is ignored.

10

Coordinate transformation

10.1 Coordinate transformation cycles

10.1.1 Fundamentals

Once a contour has been programmed, the control can execute it on the workpiece at various locations and in different sizes by using cycles for coordinate transformation.

Effectiveness of coordinate transformations

Beginning of effect: A coordinate transformation takes effect as soon as it is defined —it is not called separately. It remains in effect until it is changed or canceled.

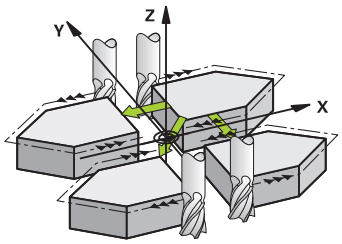
Reset coordinate transformation:

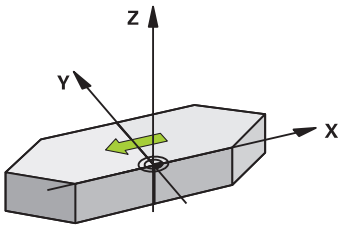
- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

10.1.2 Cycle 8 MIRRORING

ISO programming
G28

Application





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

Mirroring takes effect as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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 "jumps" accordingly.

Reset

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

Related topics

- Mirroring with **TRANS MIRROR**
  Further information: Programming and Testing User's Manual

Notes

- This cycle can be executed only 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
	Mirror image axis? Enter the axes 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. Entry: **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

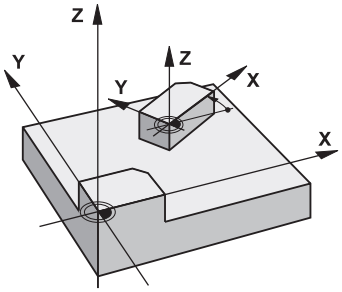
Example

```
11 CYCL DEF 8.0 MIRRORING
12 CYCL DEF 8.1 X Y Z
```

10.1.3 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 takes effect as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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°.

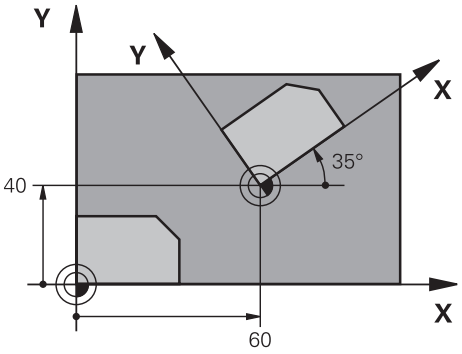
Related topics

- Rotation with **TRANS ROTATION**
  Further information: Programming and Testing User's Manual

Notes

- This cycle can be executed only 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**

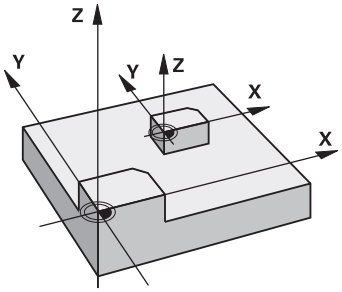
Example

```
11 CYCL DEF 10.0 ROTATION
12 CYCL DEF 10.1  ROT+35
```

10.1.4 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 takes effect as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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 be executed only in the **FUNCTION MODE MILL** machining mode.

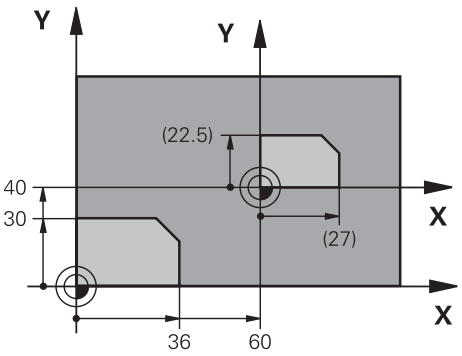
Reset

Program Cycle **11 SCALING FACTOR** again and specify a scaling factor of 1.

Related topics

- Scaling with **TRANS SCALE**
  Further information: Programming and Testing User's Manual

Cycle parameters

Help graphic	Parameter
	**Factor?** Enter the scaling factor SCL. The control multiplies the coordinates and radii with SCL. Input: **0.000001...99.999999**

Example

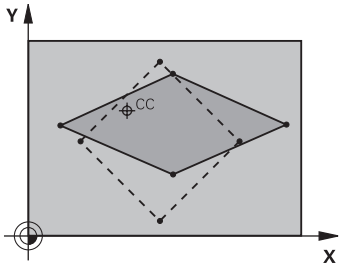
```
11 CYCL DEF 11.0 SCALING FACTOR
12 CYCL DEF 11.1 SCL 0.75
```

10.1.5 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 takes effect as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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.

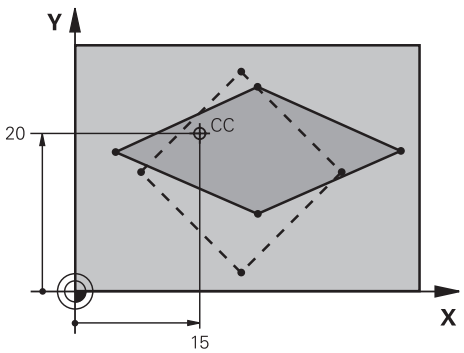
Notes

- This cycle can be executed only 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 the action bar. 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**

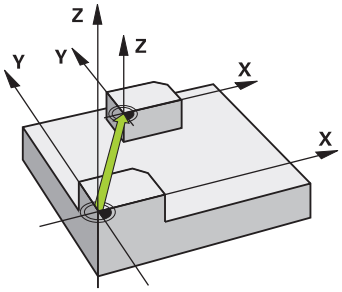
Example

```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

10.1.6 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 **Program Run** the control shows the active preset number behind the preset symbol in the **Positions** workspace.

Related topics

- Activate the preset
  Further information: Programming and Testing User's Manual
- Copy the preset
  Further information: Programming and Testing User's Manual
- Correct the preset
  Further information: Programming and Testing User's Manual
- Setting and activating presets
  Further information: User's Manual for Setup and Program Run

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- Before activating a preset, check whether all columns contain values.
- For undefined columns, enter values (e.g., **0**)
- As an alternative, have the machine manufacturer define **0** as the default value for the columns

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- 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** operating mode.
- Cycle **247** is also in effect in the simulation.

Cycle parameters

Help graphic	Parameter
	Number for preset? Enter the number of the desired preset from the preset table. Alternatively, you can use the button with the preset symbol in the action bar to directly select the desired preset from the preset table. Input: **0...65535**

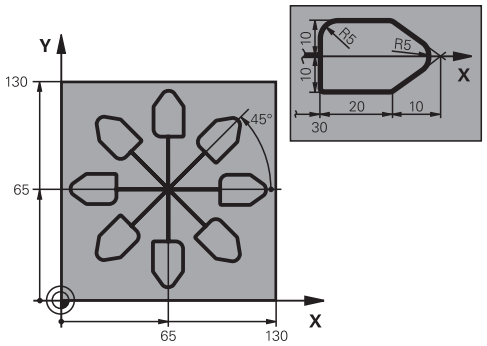
Example

```
11 CYCL DEF 247 PRESETTING ~
    Q339=+4                 ;PRESET NUMBER
```

10.1.7 Example: Coordinate conversion 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 of program run
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
```

11

Control functions

11.1 Cycles with control function

11.1.1 Overview

Cycle			Further information
9	**DWELL TIME** ■ Delay execution by the programmed dwell time	**DEF**-active	Page 431
13	**ORIENTATION** ■ Rotate spindle to a specific angle	**DEF**-active	Page 432
32	**TOLERANCE** ■ Program the permissible contour deviation for jerk-free machining operations	**DEF**-active	Page 434
239	**ASCERTAIN THE LOAD** (#143 / #2-22-1) ■ Measure and adapt the current inertia of loading, the current friction forces and the maximum axis acceleration automatically ■ Reset feedforward and control parameters	**DEF**-active	Page 439

11.1.2 Conditional stops in cycles with control functions

If your machine has an override controller, you can activate conditional stops during program run. If you activate the conditional stops with the **In cycle call** selection, the control will **not** interrupt program run of the following cycles:

- Cycle **9 DWELL TIME**
- Cycle **13 ORIENTATION**
- Cycle **32 TOLERANCE**
- Cycle **239 ASCERTAIN THE LOAD** (#143 / #2-22-1)

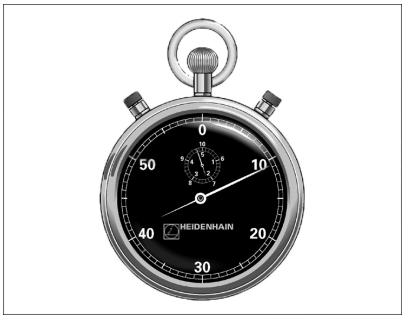
Further information: User's Manual for Setup and Program Run

11.1.3 Cycle 9 DWELL TIME

ISO programming
G4

Application

You can execute this cycle in the following operating modes: **FUNCTION MODE MILL**.



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 takes effect as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Related topics

- Dwell time with **FUNCTION FEED DWELL**
  Further information: Programming and Testing User's Manual
- Dwell time with **FUNCTION DWELL**
  Further information: Programming and Testing User's Manual

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

Example

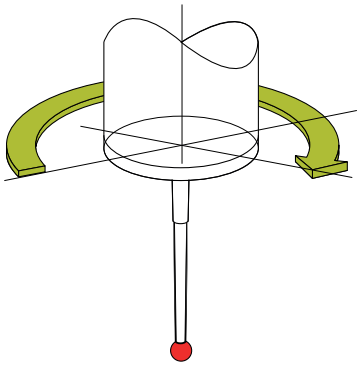
```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

11.1.4 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 numerical 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 transceiver window of HEIDENHAIN 3D 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 manufacturer.

Notes

- You can execute this cycle in the following operating modes: **FUNCTION MODE MILL**.
- 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
	Orientation angle Enter the angle relative to the angle reference axis of the working plane. Input: **0...360**

Example

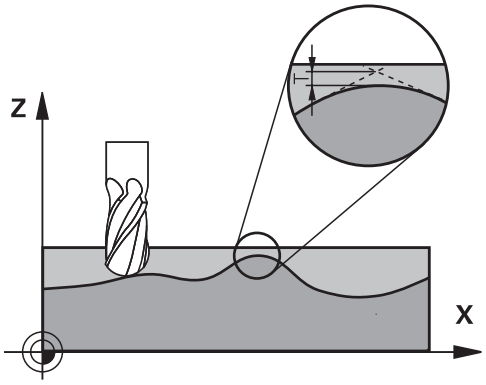
```
11 CYCL DEF 13.0 ORIENTATION
12 CYCL DEF 13.1 ANGLE180
```

11.1.5 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 corrected 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 manufacturer 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.

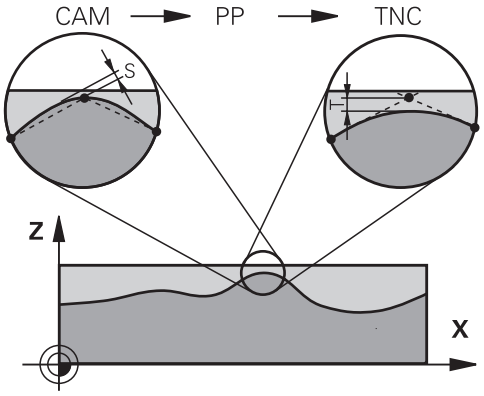
Reset

The control resets Cycle **32** if you do one of the following:

- Redefine Cycle **32** and confirm the dialog prompt for the **tolerance value** with **NO ENT**
- Select a new NC program

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% L of the CAM chord error.

Related topics

- Working with CAM-generated NC programs
  Further information: Programming and Testing User's Manual

Notes

NOTICE

Danger of collision!

To shorten the machining time, you can define greater path deviations for **FMAX** in Cycle **32 TOLERANCE**. With greater path deviations, collisions or damage to the workpiece are possible.

- ▶ Watch out for possible collisions!
- ▶ Define the **T-FMAX** parameter in accordance with the machining operation

NOTICE

Caution: Danger to the tool and workpiece!

If you combine Cycle **32 TOLERANCE** with other machine-specific tuning or optimization cycles, unexpected reactions are possible. The combination might, for example, inadvertently overwrite individual cycle parameters and thus lead to undesired machine behavior. In this case, tool and workpiece damage may result during subsequent machining operations.

- Use only a single tuning or optimization cycle
- Deactivate active cycles, if necessary, in order to avoid overlaps

- You can execute this cycle in the following operating modes: **FUNCTION MODE MILL**.
- Cycle **32** is DEF-active which means that it takes effect as soon as it is defined in the NC program.
- In a program with millimeters set as unit of measure, the control interprets the tolerance values entered in **T** and **T-FMAX** as millimeters. In an inch program, it interprets them as inches.
- As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine manufacturer).
- 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 **32**, 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 ~ K x L x TA K = 0.0175 [1/°]
  Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

Sample formula for a toroid cutter:

When machining with a toroid cutter, the angle tolerance is very important.

$$Tw = \frac{180}{\pi * R} T_{32}$$

T_w: Angle tolerance in degrees
π: Circular constant (pi)
R: Major radius of the torus in mm
T_{32}: Machining tolerance in mm

Cycle parameters

Help graphic	Parameter
	T Tolerance of contour deviation Permitted contour deviation in mm or inch **>0**: The control uses the maximum permitted deviation you have specified. **0**: The control uses a value configured by the machine manufacturer. When skipping this parameter with **NO ENT**, the control uses a value configured by the machine manufacturer. Input: **0...10**
	HSC-MODE: Finishing=0, Roughing=1 Activate filter: **0**: Milling with increased contour accuracy. The control uses internally defined finishing filter settings. **1**: Milling with increased feed rate. The control uses internally defined roughing filter settings. Input: **0**, **1**
	TA Tolerance for rotary axes Permissible position error of rotary axes in degrees with active **M128** (**FUNCTION TCPM**). The control always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for NC programs for more than one axis by entering a large tolerance value (e.g., 10°), because the control does not always have to position the rotary axis at the given nominal position. The tool orientation (position of the rotary axis with respect to the workpiece surface) will be adjusted. The position at the **T**ool **C**enter **P**oint (**TCP**) will be corrected automatically. For example, with a spherical cutter measured in its center and programmed based on the center path, there will be no adverse effects on the contour. **>0**: The control uses the maximum permitted deviation you have programmed. **0**: The control uses a value configured by the machine manufacturer. When skipping this parameter with **NO ENT**, the control uses a value configured by the machine manufacturer. Input: **0...10**

Help graphic	Parameter
	T-FMAX Tolerance of path deviation at rapid traverse Permitted path deviation in rapid traverse **FMAX** in mm or inches **>0**: In positioning blocks with **FMAX**, the control uses the maximum permitted deviation you have specified. **0**: In positioning blocks with **FMAX**, the control uses the same tolerance as in the **T** parameter. When removing this parameter with **NO ENT**, the control uses the same tolerance as in the **T** parameter. Input: **0...10**

Example

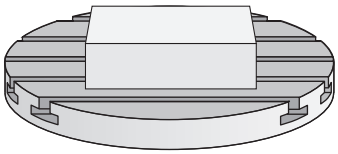
```
11 CYCL DEF 32.0 TOLERANCE
12 CYCL DEF 32.1 T0.02
13 CYCL DEF 32.2 HSC-MODE:1 TA5
13 CYCL DEF 32.3 T-FMAX2
```

11.1.6 Cycle 239 ASCERTAIN THE LOAD (#143 / #2-22-1)

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 static friction of the table axes. With the Load Adapt. Contr. (#143 / #2-22-1) software option and Cycle **239 ASCERTAIN THE LOAD**, the control is able to automatically determine and adjust the current mass inertia of the load, the current 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 manufacturer 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 manufacturer.

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 The axes will not be moved physically.
2 The control resets 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 one or more axes might be moved. 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 manufacturer.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the determined 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

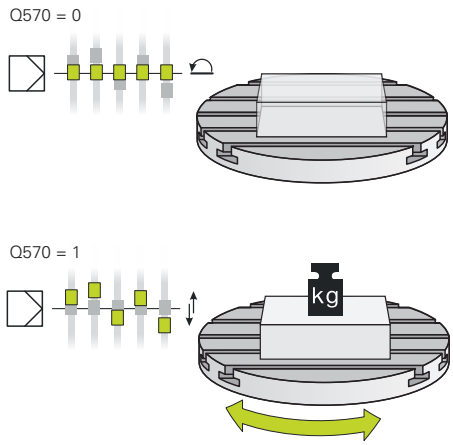
Danger of collision!

This cycle may perform extensive movements in one or more axes at rapid traverse! There is a danger of collision!

- Contact your machine manufacturer 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 manufacturer 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 mode.
- Cycle **239** takes effect 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
	**Q570 Load (0 = Delete/1 = Ascertain)?** 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: **0**: Reset LAC; the values most recently ascertained by the control are reset, and the control uses load-independent feedforward and controller parameters **1**: Perform a weighing run; the control moves the axes and thus ascertains the feedforward and controller parameters depending on the current load. The values ascertained are activated immediately. Input: **0**, **1**

Example

```
11 CYCL DEF 239  ASCERTAIN THE LOAD ~
    Q570=+0                    ;LOAD ASCERTATION
```

12

Component monitoring (#155 / #5-02-1)

12.1 Cycle 238 MEASURE MACHINE STATUS (#155 / #5-02-1)

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 the software option Component Monitoring (#155 / #5-02-1) 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 manufacturer. The machine manufacturer can read and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

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

Related topics

- Component monitoring with **MONITORING HEATMAP** (#155 / #5-02-1)
  Further information: Programming and Testing User's Manual

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 manufacturer 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** 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 and Program Run

Your machine manufacturer defines in detail how the axes will move.

Notes

Cycle **238 MEASURE MACHINE STATUS** can be hidden with the optional machine parameter **hideCoMo** (no. 128904).

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 mode.
- 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; then press the **NC Start** key to run the cycle again.
- If your machine has an override controller, you can activate conditional stops during program run. When activating the conditional stops by selecting **In cycle call**, the control will not interrupt the program run at Cycle **238 MEASURE MACHINE STATUS**.

12.1.1 Cycle parameters

Help graphic	Parameter
	Q570 Mode (0=test/1=measure)? Define whether the control will perform a measurement of the machine status in test mode or in measurement mode: **0**: No measured data will be generated. You can control the axis movements with the feed rate and rapid traverse potentiometers **1**: This mode will generate measured data. You **cannot** control the axis movements with the feed rate and rapid traverse potentiometers Input: **0**, **1**

Example

```
11 CYCL DEF 238  MEASURE MACHINE STATUS ~
    Q570=+0                    ;TEST OF MOTIONS
```

13

Multiple-axis machining

13.1 Cycles for cylinder surface machining

13.1.1 Overview

Cycle		Call	Further information
27	**CYLINDER SURFACE** (#8 / #1-01-1) ■ Milling of guide slots on the cylinder surface ■ Slot width is equal to tool radius	**CALL**-active	Page 447
28	**CYLINDRICAL SURFACE SLOT** (#8 / #1-01-1) ■ Milling of guide slots on the cylinder surface ■ Input of the slot width	**CALL**-active	Page 450
29	**CYL SURFACE RIDGE** (#8 / #1-01-1) ■ Milling of a ridge on the cylinder surface ■ Input of the ridge width	**CALL**-active	Page 454
39	**CYL. SURFACE CONTOUR** (#8 / #1-01-1) ■ Milling of a contour on the cylinder surface	**CALL**-active	Page 458

13.1.2 Conditional stops in cylinder surface cycles

If your machine has an override controller, you can activate conditional stops during program run. If you activate conditional stops with the **In cycle call** selection, the control interrupts at the following breakpoints:

The control stops before the first movement.

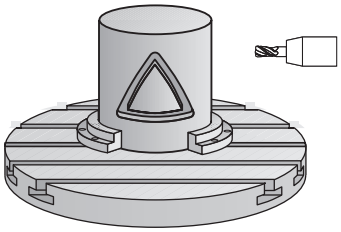
Further information: User's Manual for Setup and Program Run

13.1.3 Cycle 27 CYLINDER SURFACE (#8 / #1-01-1)

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 coordinate data of the unrolled cylinder surface (X coordinates), which define the position of the rotary table, can be entered as desired either in degrees or in mm (or inches) (**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

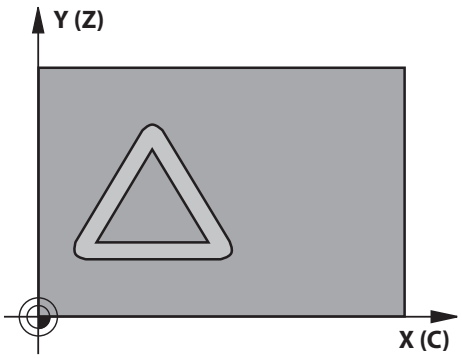
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular 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

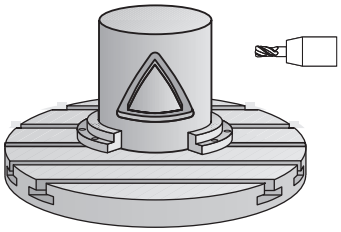
```
11 CYCL DEF 27 CYLINDER SURFACE ~
    Q1=-20          ;MILLING DEPTH ~
    Q3=+0           ;ALLOWANCE FOR SIDE ~
    Q6=+0           ;SET-UP CLEARANCE ~
    Q10=-5          ;PLUNGING DEPTH ~
    Q11=+150        ;FEED RATE FOR PLNGNG ~
    Q12=+500        ;FEED RATE F. ROUGHNG ~
    Q16=+0          ;RADIUS ~
    Q17=+0          ;CYLINDER COORDINATES IN DEGREES
```

13.1.4 Cycle 28 CYLINDRICAL SURFACE SLOT (#8 / #1-01-1)

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 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. There is a danger of collision!

- Control the traversing movements of the machine
- In the **Simulation** workspace of the **Editor** operating mode, check the end position of the tool after the cycle
- After the cycle, program absolute coordinates (no incremental coordinates)

- This cycle can be executed only 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.

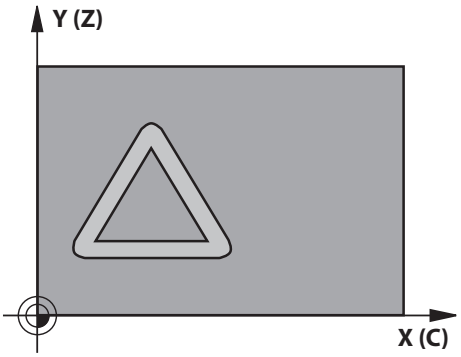
Notes on programming

- 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**

Help graphic	Parameter
	Q20 Slot width? Width of the slot to be machined Input: **-99999.9999...+99999.9999**
	Q21 Tolerance? (optional) 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

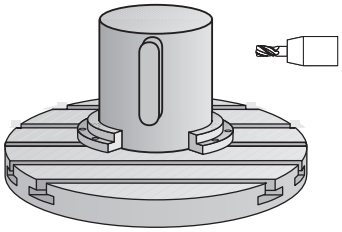
```
11 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~
    Q1=-20          ;MILLING DEPTH ~
    Q3=+0           ;ALLOWANCE FOR SIDE ~
    Q6=+2           ;SET-UP CLEARANCE ~
    Q10=-5          ;PLUNGING DEPTH ~
    Q11=+150        ;FEED RATE FOR PLNGNG ~
    Q12=+500        ;FEED RATE F. ROUGHNG ~
    Q16=+0          ;RADIUS ~
    Q17=+0          ;CYLINDER COORDINATES IN DEGREES ~
    Q20=+0          ;SLOT WIDTH ~
    Q21=+0          ;TOLERANCE
```

13.1.5 Cycle 29 CYL SURFACE RIDGE (#8 / #1-01-1)

ISO programming
G129

Application

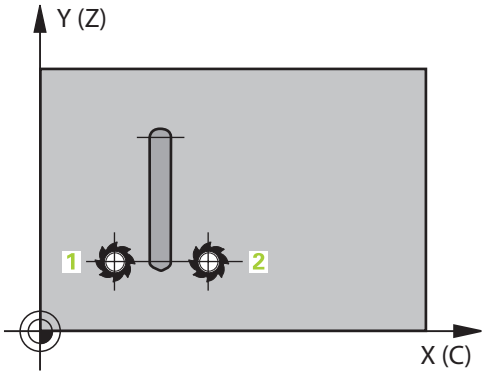
Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



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 be executed only 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
	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 ridge wall. The finishing allowance increases the ridge 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**

Help graphic	Parameter
	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 Ridge width? Width of the ridge to be machined Input: **-99999.9999...+99999.9999**

Example

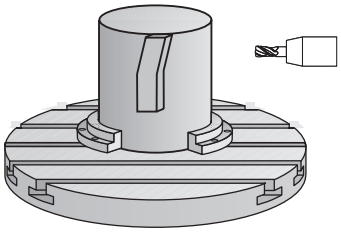
```
11 CYCL DEF 29 CYL SURFACE RIDGE ~
    Q1=-20          ;MILLING DEPTH ~
    Q3=+0           ;ALLOWANCE FOR SIDE ~
    Q6=+2           ;SET-UP CLEARANCE ~
    Q10=-5          ;PLUNGING DEPTH ~
    Q11=+150        ;FEED RATE FOR PLNGNG ~
    Q12=+500        ;FEED RATE F. ROUGHNG ~
    Q16=+0          ;RADIUS ~
    Q17=+0          ;CYLINDER COORDINATES IN DEGREES ~
    Q20=+0          ;RIDGE WIDTH
```

13.1.6 Cycle 39 CYL. SURFACE CONTOUR (#8 / #1-01-1)

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 be executed only 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
	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**

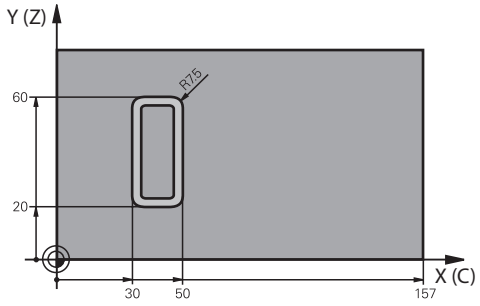
Example

```
11 CYCL DEF 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                ;CYLINDER COORDINATES IN DEGREES
```

13.1.7 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

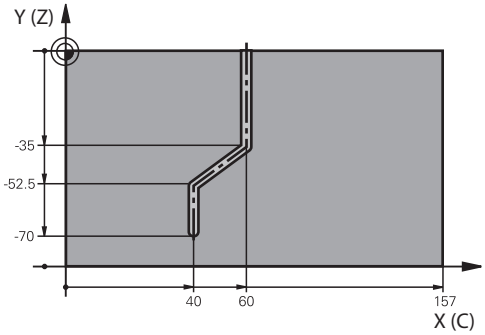


0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call (diameter: 7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-7 ;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 ;CYLINDER COORDINATES IN MM/ INCH	
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 run
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 LABEL1	
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 ;CYLINDER COORDINATES IN MM/ INCH ~	
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 run
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	

14

Programming with variables

14.1 Program defaults for cycles

14.1.1 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 globally effective for all cycles used in the NC program. In the respective cycle you then use **PREDEF** to simply reference the value defined at the beginning of the program.

The following **GLOBAL DEF** functions are available:

Cycle		Call	Further information
100	**GENERAL** Definition of generally valid cycle parameters ■ **Q200 SET-UP CLEARANCE** ■ **Q204 2ND SET-UP CLEARANCE** ■ **Q253 F PRE-POSITIONING** ■ **Q208 RETRACTION FEED RATE**	**DEF**-active	Page 468
105	**DRILLING** Definition of specific drilling cycle parameters ■ **Q256 DIST FOR CHIP BRKNG** ■ **Q210 DWELL TIME AT TOP** ■ **Q211 DWELL TIME AT DEPTH**	**DEF**-active	Page 469
110	**POCKET MILLING** Definition of specific pocket-milling cycle parameters ■ **Q370 TOOL PATH OVERLAP** ■ **Q351 CLIMB OR UP-CUT** ■ **Q366 PLUNGE**	**DEF**-active	Page 470
111	**CONTOUR MILLING** Definition of specific contour-milling cycle parameters ■ **Q2 TOOL PATH OVERLAP** ■ **Q6 SET-UP CLEARANCE** ■ **Q7 CLEARANCE HEIGHT** ■ **Q9 ROTATIONAL DIRECTION**	**DEF**-active	Page 471
125	**POSITIONING** Definition of the positioning behavior with **CYCL CALL PAT** ■ **Q345 SELECT POS. HEIGHT**	**DEF**-active	Page 471

14.1.2 Entering GLOBAL DEF definitions

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **GLOBAL DEF**
- ▶ Select the desired **GLOBAL DEF** function (e.g., **100 GENERAL**)
- ▶ Enter the required definitions

14.1.3 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:

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select and define **GLOBAL DEF**
- Select **Insert NC function** again
- Select the desired cycle (e.g., **200 DRILLING**)
- > If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

- Select **PREDEF**
- > The control then 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. There is a danger of collision!

- Make sure to use **GLOBAL DEF** carefully. Simulate your program before executing it
- If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

14.1.4 Global data valid everywhere

Parameters valid for all machining cycles **2xx** and the touch probe cycles **451**, **452**

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

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
```

14.1.5 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
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0.1...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**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**

Example

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

14.1.6 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
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.1...1999**
	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**
	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**: 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 **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. The reciprocation length depends on the plunging angle. As a minimum value the control uses twice the tool diameter. Input: **0**, **1**, **2**

Example

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

14.1.7 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
	Q2 Path overlap factor? **Q2** x tool radius = stepover factor k Input: **0.0001...1.9999**
	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). This value has an absolute effect. Input: **-99999.9999...+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 GLOBAL DEF 111 CONTOUR MILLING ~
    Q2=+1                ;TOOL PATH OVERLAP ~
    Q6=+2                ;SET-UP CLEARANCE ~
    Q7=+50               ;CLEARANCE HEIGHT ~
    Q9=+1                ;ROTATIONAL DIRECTION
```

14.1.8 Global data for positioning behavior

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

Help graphic	Parameter
	Q345 Select positioning height (0/1) 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: **0**, **1**

Example

```
11 GLOBAL DEF 125 POSITIONING ~
    Q345=+1              ;SELECT POS. HEIGHT
```

15

User aids

15.1 OCM cutting data calculator (#167 / #1-02-1)

15.1.1 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 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.

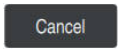
15.1.2 Operation

Opening the cutting data calculator

- Select cycle **272 OCM ROUGHING**
- Select **OCM cutting data calculator** in the action bar

Closing the cutting data calculator

- Select **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

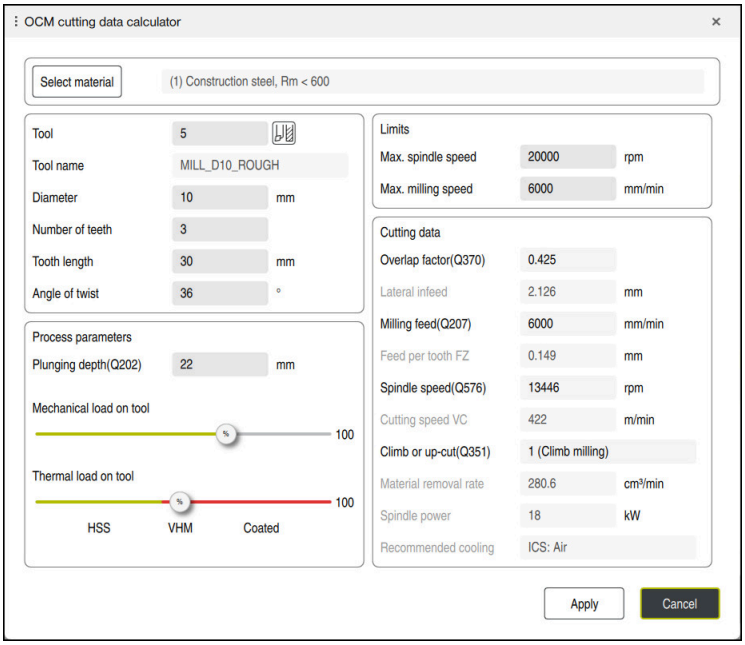
- Select **Cancel**
- > The current entries are not stored.
- > The control does not apply any values to the cycle.

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)

If you use the OCM cutting data calculator, then do not subsequently edit these parameters in the cycle.

15.1.3 Fillable form

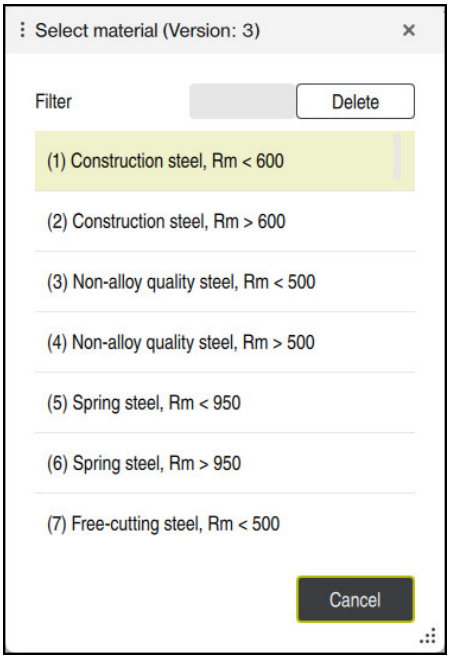


The control uses various colors and symbols in the fillable form:

- Dark gray background: entry required
- Red border of input boxes and information symbols: missing or incorrect entry
- Gray background: no entry possible

- The input field of the workpiece material is highlighted in gray. You can only select it through the selection list. The tool can also be selected through the tool table.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

Workpiece material



To select the workpiece material:

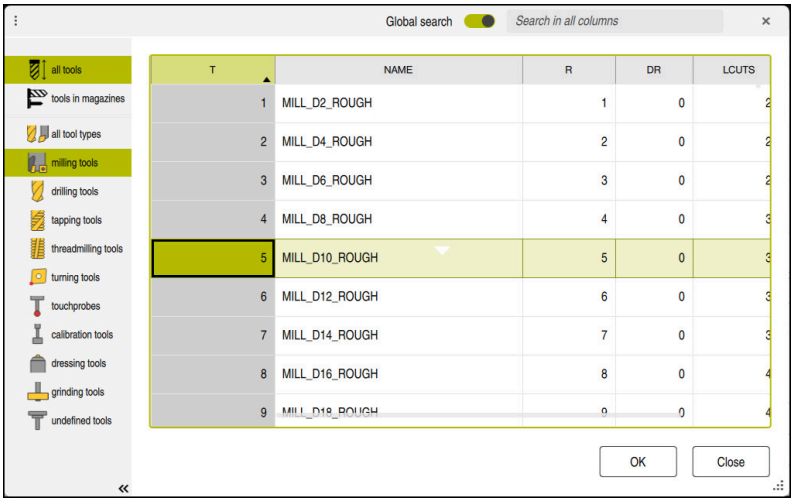
- Select the **Select material** button
- > The control opens a selection list with various types of steel, aluminum, and titanium.
- Select the workpiece material

  or
- Enter a search term in the filter mask
- > The control displays the materials or material groups that were found. Use the **Delete** button to return to the original selection list.

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
- 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:

- Select the **Select the tool** button

> The control opens the active tool table **tool.t**.

- Select the tool

  or

- Enter a tool name or number in the search field
- Confirm with **OK**

> The control applies the **Diameter**, the **Number of teeth** and the **Tooth length** from the **tool.t** table.

- Define the **Angle of twist**

To select the tool:

- Enter the **Diameter**
- Define the **Number of teeth**
- Enter the **Tooth length**
- Define the **Angle of twist**

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

Programming and operating notes:
- You can modify the values of the **Diameter**, the **Number of teeth** and the **Tooth length** at any time. The modified value is **not** written to the **tool.t** tool table!
- 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: **1...99999**
Max. milling speed	Maximum milling speed (feed rate) in mm/min permitted by the machine and the clamping situation: Input: **1...99999**

Process parameters

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

Input dialog	Description
Plunging depth(Q202)	Plunging depth (>0 mm to [6 times the tool diameter]) The value from cycle parameter **Q202** is applied when starting the OCM cutting data calculator. Input: **0.001...99999.999**
Mechanical load on tool	Slider for selection of the mechanical load (the value is normally between 70% and 100%) Input: **0%...150%**
Thermal load on tool	Slider for selection of the thermal load Set the slider according to the thermal wear-resistance (coating) of your tool. ■ HSS: low thermal wear-resistance ■ VHM (uncoated or normally-coated solid carbide milling cutters): medium thermal wear-resistance ■ Coated (fully-coated solid carbide milling cutters): high thermal wear-resistance ■ The slider is effective only in the range with a green background. This limiting depends on the maximum spindle speed, the maximum feed rate, and the selected material. ■ If the slider is in the red range, the control will use the maximum permissible value. Input: **0%...200%**

Further information: "Process parameters ", Page 481

Cutting data

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

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

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

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^3/min
- Spindle power in kW
- Recommended cooling

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

15.1.4 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>=16 mm), a very high level of spindle power can be necessary.

For the theoretically expected 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.

15.1.5 Achieving an optimum result

If the Cutting data determined 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	-	■ Reduce the plunging depth ■ Use a tool with a lower angle of twist
Excessive radial force on spindle bearing	Decrease	-	

Index

A

About the product ... 39
About the User's Manual ... 25
Additional documentation ... 28
Application
  Help ... 33
  Start/Login ... 55

C

Centering ... 218
Contact ... 37
Context-sensitive help ... 35
Contour call
  CONTOUR DEF ... 107, 111
  Cycle 14 Contour ... 106
Contour formula
  Complex ... 111
  Simple ... 107
Control's user interface ... 53
Coordinate transformation
  Cycle Axis-specific scaling factor ... 423
  Mirroring cycle ... 417
  Rotation cycle ... 419
  Scaling cycle ... 421
Countersinking
  Back boring ... 214
Cylindrical surface cycles
  Contour ... 458
  Cylindrical surface ... 447
  Ridge ... 454
  Slot ... 450

D

Drilling
  Bore milling ... 201
  Drilling ... 179
  Reaming ... 183, 185
  Single-lip deep hole drilling ... 206
  Universal deep-hole drilling ... 194
  Universal drilling ... 189
Drilling, centering and thread cycles
  Countersinking and centering ... 214
  Drilling ... 179
  Tapping ... 221
  Thread milling ... 235
Dwell time ... 431

E

Engraving ... 407
Export version ... 45

F

First steps ... 57
  Programming ... 58

G

GLOBAL DEF ... 466
GUI ... 53

I

Integrated product aid
  TNCguide ... 32

L

Licensing terms ... 52

M

Milling contour
  Superimposing contours ... 102
Milling cycles
  Engraving ... 407
  Milling contours with OCM cycles ... 352
  Milling contours with SL cycles ... 310
  Milling planes ... 390
  Milling pockets ... 264
  Milling studs ... 290
Milling planes
  Extended face milling ... 397
  Face milling ... 390
Milling pockets
  Circular pocket ... 271
  Rectangular pocket ... 264
Milling slots
  Circular slot ... 283
  Slot milling ... 277
Milling studs
  Circular stud ... 296
  Polygon stud ... 301
  Rectangular stud ... 290
Monitoring
  Ascertaining the load ... 439
  Measure machine condition ... 442

N

Notes, types of ... 29

O

OCM
  Cutting data calculator ... 474
OCM cycles
  Chamfering ... 374
  Contour data ... 360
  Figure cycles ... 150
  Finishing floor ... 368
  Finishing side ... 371
  Roughing ... 363
OCM figures
  Circle ... 157
  Circle boundary ... 172
  Circular slot ... 163
  Polygon ... 166
  Rectangle ... 153
  Rectangle boundary ... 170
  Slot / ridge ... 160
Operating mode
  Machine ... 55
  Manual ... 55
  Overview ... 55
  Start ... 55
Operational cycle parameter ... 85

P

Pattern cycles
  Circle ... 136
  DataMatrix code ... 142
  Lines ... 139
PATTERN DEF
  Call ... 124
  Programming ... 124
Pattern definition
  Cycles ... 134
  PATTERN DEF ... 123
  Point table ... 120
Pattern definition with PATTERN DEF
  frames ... 128
  full circle ... 130
  patterns ... 127
  pitch circle ... 131
  Point ... 125
Place of operation ... 42
Point table
  Cycle call ... 121
  Selecting ... 121
Program call
  Cycle PGM CALL ... 98
Program examples
  Pattern cycles ... 148
  PATTERN DEF ... 132
Programming example
  Milling a pocket and a stud ... 307
Programming examples
  Coordinate transformation ... 426
  Cylinder surface ... 461
  OCM cycles ... 378
  SL cycles ... 346
Programming technique ... 97
Programming with variables ... 465
Proper and intended operation ... 40

S

Safety precaution ... 43
  Content ... 29
Selection function
  NC program as contour ... 114
  NC program as cycle ... 92
SEL PATTERN ... 121
Setting a preset ... 424
SL Cycles

3-D contour train..... 341
Contour data..... 313
Contour train..... 330
Contour train data..... 328
Floor finishing..... 321
Fundamentals..... 310
Pilot drilling..... 315
Roughing..... 317
Side finishing..... 324
Superimposed contours..... 117
Trochoidal milling of contour slot..... 335
Software number..... 45
Software option..... 46
Spindle orientation..... 432

T

Tapping
With chip breaking..... 231
With floating tap holder..... 224
Without floating tap holder.... 227
Target group..... 26
Thread cutting..... 221
Thread milling
Fundamentals..... 235
Helical thread drilling/milling. 251
Inside..... 237
Outside..... 255
Thread drilling/milling..... 246
Thread milling/countersinking..... 241
TNCguide..... 33
Tolerance..... 434

U

User Aids..... 473
User interface of the control..... 53
User's Manuals..... 27

V

Variable..... 465

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com
The Information Site for HEIDENHAIN Controls

Klartext App
Klartext on your mobile device

Google Play Store

Apple App Store

Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.

For more details on touch probes and vision systems:
www.heidenhain.com/products/touch-probes-and-vision-systems

Original document
1410289-22 · Ver02 · 10/2025 · H · Printed in Germany