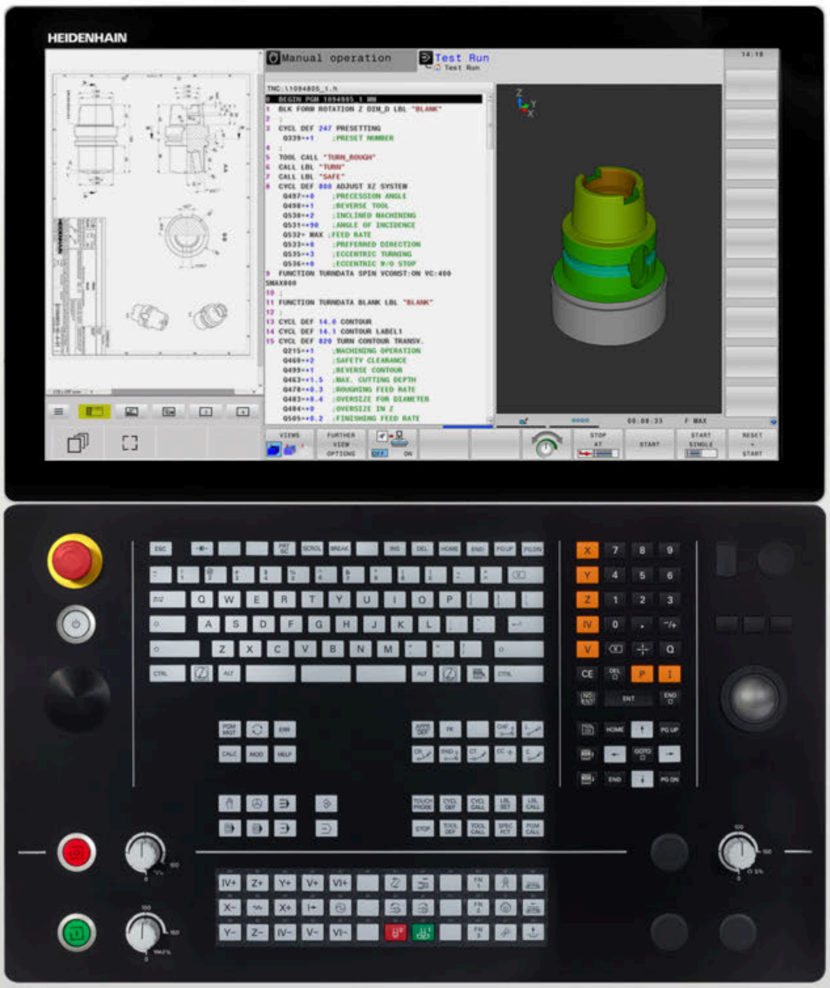
HEIDENHAIN



TNC 640

User's Manual
Programming of
Machining Cycles

NC Software
34059x-18

English (en)
10/2023

Table of contents

1

Fundamentals

1.1 About this manual

Safety precautions

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

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

DANGER

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

WARNING

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

CAUTION

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

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in property damage**.

Sequence of information in precautionary statements

All precautionary statements 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.

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

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

tnc-userdoc@heidenhain.de

1.2 Control model, software, and features

This manual describes programming functions provided by our controls with the following NC software numbers and later.

Control model	NC software number
TNC 640	340590-18
TNC 640 E	340591-18
TNC 640 programming station	340595-18

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

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

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

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

- Tool measurement with the TT

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

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

User's Manual:

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

ID of User's Manual for Programming of Measuring Cycles for Workpieces and Tools: 1303409-xx

User's Manual:

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

Klartext Programming User’s Manual ID: 892903-xx

ISO Programming User’s Manual ID: 892909-xx

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

Software options

The TNC 640 features various software options, each of which can be enabled separately by your machine manufacturer. The respective options provide the functions listed below:

Additional Axis (option 0 to option 7)	
Additional axis	Additional control loops 1 to 8
Advanced Function Set 1 (option 8)	
Advanced functions (set 1)	**Machining with rotary tables** ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute **Coordinate conversions:** Tilting the working plane **Interpolation:** Circular in 3 axes with tilted working plane
Advanced Function Set 2 (option 9)	
Advanced functions (set 2) Subject to export license	**3D machining:** ■ 3D tool compensation through surface-normal vectors ■ Changing the swivel-head angle with the electronic handwheel during program run without affecting the position of the tool tip (TCPM = **T**ool **C**enter **P**oint **M**anagement) ■ Keeping the tool normal to the contour ■ Tool radius compensation normal to the tool direction ■ Manual traverse in the active tool-axis system **Interpolation:** Linear in more than 4 axes (subject to export license)
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component
DCM Collision (option 40)	
Dynamic Collision Monitoring	■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Collision monitoring in the Test Run mode ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements
CAD Import (option 42)	
CAD import	■ Support for DXF, STL, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs

Global PGM Settings – GPS (option 44)	
Global program settings	■ Superimposition of coordinate transformations during program run ■ Handwheel superimpositioning
Adaptive Feed Control – AFC (option 45)	
Adaptive Feed Control	**Milling:** ■ Recording the actual spindle power by means of a teach-in cut ■ Defining the limits of automatic feed rate control ■ Fully automatic feed control during program run **Turning (option 50):** ■ Cutting force monitoring during machining
KinematicsOpt (option 48)	
Optimizing the machine kinematics	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics
Turning (option 50)	
Milling and turning modes	**Functions:** ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning-specific contour elements ■ Turning cycles ■ Eccentric Turning ■ Cycle **880** or **G880 GEAR HOBBING** (options 50 and 131)
KinematicsComp (option 52)	
Three-dimensional compensation	Compensation of position and component errors
OPC UA NC Server (1 to 6) (options 56 to 61)	
Standardized interface	The OPC UA NC Server provides a standardized interface (**OPC UA**) for external access to the control's data and functions. These software options allow you to create up to six parallel client connections.
3D-ToolComp (option 92)	
3D tool radius compensation depending on the tool's contact angle Subject to export license	■ Compensate the deviation of the tool radius depending on the tool's contact angle ■ Compensation values in a separate compensation value table ■ Prerequisite: Working with surface-normal vectors (**LN** blocks option 9)

Extended Tool Management (option 93)	
Extended tool management	Python-based expansion of tool management ■ Program-specific or pallet-specific usage sequence of all tools ■ Program-specific or pallet-specific tooling list of all tools

Advanced Spindle Interpolation (option 96)	
Interpolating spindle	**Interpolation turning:** ■ Cycle **291 COUPLG.TURNG.INTERP.** (ISO: **G291**) ■ Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**)

Spindle Synchronism (option 131)	
Spindle synchronization	■ Synchronization of milling spindle and turning spindle ■ Cycle **880 GEAR HOBBING** (ISO: **G880**) (options 50 and 131)

Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface

Synchronizing Functions (option 135)	
Synchronization functions	**Real Time Coupling (RTC):** Coupling of axes

Cross Talk Compensation – CTC (option 141)	
Compensation of axis couplings	■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of the TCP (**T**ool **C**enter **P**oint)

Position Adaptive Control – PAC (option 142)	
Adaptive position control	■ Adaptation of the control parameters depending on the position of the axes in the working space ■ Adaptation of the control parameters depending on the speed or acceleration of an axis

Load Adaptive Control – LAC (option 143)	
Adaptive load control	■ Automatic determination of workpiece weight and frictional forces ■ Adaptation of the control parameters depending on the current mass of the workpiece

Active Chatter Control – ACC (option 145)	
Active chatter control	Fully automatic function for chatter control during machining

Machine Vibration Control – MVC (option 146)	
Vibration damping for machines	Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ Active Vibration Damping (**AVD**) ■ Frequency Shaping Control (**FSC**)

CAD Model Optimizer (option 152)	
Optimization of CAD models	Conversion and optimization of CAD models ■ Fixtures ■ Workpiece blank ■ Finished part
Batch Process Manager (option 154)	
Batch process manager	Planning of production orders
Component Monitoring (option 155)	
Component monitoring without external sensors	Monitoring of configured machine components for overload
Grinding (option 156)	
Jig grinding	■ Reciprocating stroke cycles ■ Cycles for dressing ■ Support of the "dressing tool" and "grinding tool" tool types
Gear Cutting (option 157)	
Machining gear systems	■ Cycle **285 DEFINE GEAR** (ISO: **G285**) ■ Cycle **286 GEAR HOBBING** (ISO: **G286**) ■ Cycle **287 GEAR SKIVING** (ISO: **G287**)
Turning v2 (option 158)	
Mill-turning version 2	■ All functions of software option 50 ■ Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** ■ Cycle **883 TURNING SIMULTANEOUS FINISHING** The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation.
Opt. contour milling (option 167)	
Optimized contour cycles	Cycles for machining any pockets and islands using trochoidal milling

Further options available

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

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

ID: 827222-xx

VTC User's Manual
All functions of the software for the VT 121 vision system are described in the **VTC User's Manual**. Please contact HEIDENHAIN if you require a copy of this User's Manual.
ID: 1322445-xx

Feature content level (upgrade functions)

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

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

Upgrade functions are identified in the manual with **FCL n,** where **n** indicates the sequential number of the feature content level.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine manufacturer or HEIDENHAIN.

Intended place of operation

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

Legal information

Legal information

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

Further information is available on the control as follows:

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

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

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

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, they are always provided at the end of the cycle definition. The section "New or modified cycle functions of software 34059x-18 " gives you an overview of the optional Q parameters that have been added in this software version. You can decide for yourself whether you would like to define optional Q parameters or delete them with the NO ENT key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing NC programs after a software update, you can add optional Q parameters in cycles where needed. The following steps describe how this is done.

Proceed as follows:

- Call the cycle definition
- Press the right cursor key until the new Q parameters are displayed
- Confirm the displayed default value

  or
- Enter a value
- To load the new Q parameter, exit the menu by pressing the right arrow key once again or by pressing **END**
- If you do not wish to load the new Q parameter, press the **NO ENT** key

Compatibility

Most NC programs created with older HEIDENHAIN contouring controls (with TNC 150 B and later) can be run with the new software version of the TNC 640. Even if new optional parameters ("Optional parameters") have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way round, if you want to run an NC program created with a new software version on an older control, you can delete the respective optional Q parameters from the cycle definition with the NO ENT key. In this way you can ensure that the NC program is downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

New or modified cycle functions of software 34059x-18

Overview of new and modified software functions

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

ID: 1322095-xx

New cycle functions with 81762x-18

- Cycle **1274 OCM CIRCULAR SLOT** (ISO: **G1274**, option 167)

  This cycle allows you to define a circular slot that is then used as a pocket or boundary for face milling in conjunction with other OCM cycles.

  Further information: "Cycle 1274 OCM CIRCULAR SLOT (option 167)", Page 382

Modified cycle functions with 81762x-18

- You can also define subcontours as **LBL** subprograms within the complex **SEL CONTOUR** contour formula.

  Further information: "SL or OCM cycles with complex contour formula", Page 426
- The machine manufacturer can hide the cycles **220 POLAR PATTERN** (ISO: **G220**) and **221 CARTESIAN PATTERN** (ISO: **G221**). We recommend using the **PATTERN DEF** function.

  Further information: "Pattern definition with PATTERN DEF", Page 62
- The input value **1** has been added to parameter **Q515 FONT** in Cycle **225 ENGRAVING** (ISO: **G225**). Use this input value to select the **LiberationSans-Regular** font.

  Further information: "Cycle 225 ENGRAVING ", Page 469
- In the following cycles, you can enter symmetrical tolerances "+-...." for the nominal dimensions:
  - Cycle **208 BORE MILLING** (ISO: **G208**)
  - **127x** (option 167) OCM standard figure cycles

  Further information: "Cycle 208 BORE MILLING ", Page 106

  Further information: "OCM standard figures", Page 368
- Cycle **287 GEAR SKIVING** (ISO: **G287**, option 157) has been extended:
  - When you program the optional parameter **Q466 OVERRUN PATH**, the control will optimize the approach and idle travel paths automatically. This will reduce machining times.
  - Two columns have been added to the prototype of the technology table:
    - **dK**: Angular offset of the workpiece in order to machine one side of the tooth flank only. This can be used to increase the surface quality.
    - **PGM**: Profile program for a custom tooth flank line, for example to realize crowning of the tooth flank.
  - After each step, the control displays the number of the current cut and the number of remaining cuts in a pop-up window.

  Further information: "Cycle 287 GEAR SKIVING (option 157)", Page 496
- The machine manufacturer can configure a deviating automatic **LIFTOFF** for Cycles **286 GEAR HOBBING** (ISO: **G286**, option 157) and **287 GEAR SKIVING** (ISO: **G287**, option 157).

  Further information: "Fundamentals for the machining of gear teeth (option 157)", Page 482

- Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) has been extended:
  - The input range of the parameter **Q497 PRECESSION ANGLE** has been extended from four to five decimal places.
  - The input range of the parameter **Q531 ANGLE OF INCIDENCE** has been extended from three to five decimal places.

  Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 541
- The control shows remaining residual material during turning cycles also with the machining operations **Q215=1** and **Q215=2**.

  Further information: "Turning cycles (option 50)", Page 530
- In the touch probe cycles **14xx**, you can enter symmetrical tolerances "+-...." for the nominal dimensions.
- Cycle **441 FAST PROBING** (ISO: **G441**) now features the parameter **Q371 TOUCH POINT REACTION**. This parameter defines the reaction of the control in cases where the stylus is not deflected.
- Using the parameter **Q400 INTERRUPTION** in Cycle **441 FAST PROBING** (ISO: **G441**), you can define whether the control will interrupt program run and display a measuring log. The parameter is effective in conjunction with the following cycles:
  - Cycle **444 PROBING IN 3-D** (ISO: **G444**)
  - Touch-probe cycles **45x** for kinematics measuring
  - Touch-probe cycles **46x** for calibrating the workpiece touch probe
  - Touch-probe cycles **14xx** for determining a workpiece misalignment and for acquiring the preset
- Cycles **451 MEASURE KINEMATICS** (ISO: **G451**, option 48) and **452 PRESET COMPENSATION** (ISO: **452**, option 48) save the measured position errors of the rotary axes in the QS parameters **QS144** to **QS146**.
- Using the optional machine parameter **maxToolLengthTT** (no. 122607), the machine manufacturer defines a maximum tool length for tool touch probe cycles.
- Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

2

Fundamentals / Overviews

2.1 Introduction

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.

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

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- Test your program before executing it

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

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

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

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

2.2 Available cycle groups

Overview of machining cycles

CYCL DEF

- Press the **CYCL DEF** key

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring and counterboring	76
DRILLING/ THREAD	Cycles for tapping, thread cutting and thread milling	130
POCKETS/ STUDS/ SLOTS	Cycles for milling pockets, studs, slots, and face milling	178
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	240
SL CYCLES	SL (Subcontour List) cycles for machining contours that consist of multiple overlapping subcontours as well as cycles for cylinder surface machining and trochoidal milling	282 337
PATTERN	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	260
TURNING	Cycles for turning and gear hobbing	530
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, determining the load, gear cycles	442
GRINDING	Cycles for grinding operations and sharpening of grinding tools	720

▷

- If required, switch to machine-specific machining cycles

  The machine manufacturer can integrate these types of machining cycles.

Overview of touch probe cycles

TOUCH PROBE

▶ Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
ROTATION	Cycles for automatic measurement and compensation of workpiece misalignment	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools
PRESET	Cycles for automatic workpiece presetting	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools
MEASURING	Cycles for automatic workpiece inspection	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools
SPECIAL CYCLES	Special cycles	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools
CALIBRATE TS	Touch probe calibration	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools
KINEMATICS	Cycles for automatic kinematics measurement	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	**Further information:** User's Manual for Programming of Measuring Cycles for Workpieces and Tools

▷

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

3

Using Fixed Cycles

3.1 Working with fixed cycles

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:

- Cycles **300** to **399**
  Machine-specific cycles that are to be defined through the **CYCLE DEF** key
- Cycles **500** to **599**
  Machine-specific touch probe cycles that are to be defined through the **TOUCH PROBE** key

NOTICE

Danger of collision!

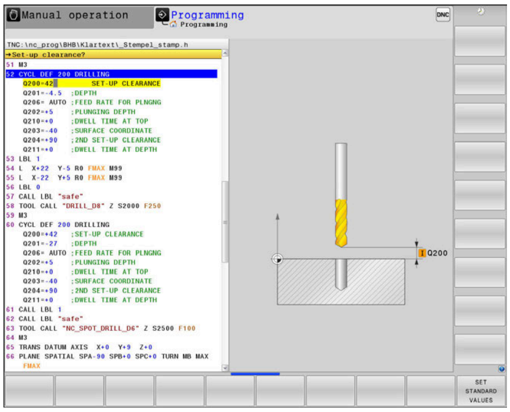
HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections and thus, undesired behavior. Danger of collision during machining!

- 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 a cycle", Page 49
Further information: User's Manual for **Klartext Programming**

Defining a cycle using soft keys

Proceed as follows:

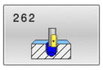


CYCL DEF

- Press the **CYCL DEF** key
- > The soft-key row shows the available groups of cycles.

DRILLING/ THREAD

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



- Select the desired cycle (e.g., Cycle **262 THREAD MILLING**)
- > The control initiates a dialog and prompts you for all required input values. At the same time, a graphic is displayed in the right half of the screen. The required parameter is highlighted.
- Enter the required parameters
- Conclude each input with the **ENT** key
- > The control closes the dialog when all required data has been entered.

NOTICE

Danger of collision!

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

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

Defining a cycle using the GOTO function

Proceed as follows:

CYCL DEF

- ▶ Press the **CYCL DEF** key
- > The soft-key row shows the available groups of cycles.

GOTO ☐

- ▶ Press the **GOTO** key
- > The control opens the smartSelect selection window with an overview of the cycles.
- ▶ Select the desired cycle with the cursor keys or the mouse.

  or
- ▶ Enter the cycle number
- ▶ Confirm each input with the **ENT** key
- > The control then initiates the cycle dialog as described above.

Example

```
11  CYCL DEF 200 DRILLING ~
    Q200=+2               ;SET-UP CLEARANCE ~
    Q201=-20              ;DEPTH ~
    Q206=+150             ;FEED RATE FOR PLNGNG ~
    Q202=+5               ;PLUNGING DEPTH ~
    Q210=+0               ;DWELL TIME AT TOP ~
    Q203=+0               ;SURFACE COORDINATE ~
    Q204=+50              ;2ND SET-UP CLEARANCE ~
    Q211=+0               ;DWELL TIME AT DEPTH ~
    Q395=+0               ;DEPTH REFERENCE
```

Calling a cycle

Requirements

Before calling a cycle, be sure to program:

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

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

The following cycles become effective automatically as soon as they have been defined in the program. You cannot and must not call them:

- Cycle **9 DWELL TIME**
- Cycle **12 PGM CALL**
- Cycle **13 ORIENTATION**
- Cycle **14 CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **32 TOLERANCE**
- Cycle **220 POLAR PATTERN**
- Cycle **221 CARTESIAN PATTERN**
- Cycle **224 DATAMATRIX CODE PATTERN**
- Cycle **238 MEASURE MACHINE STATUS**
- Cycle **239 ASCERTAIN THE LOAD**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **285 DEFINE GEAR**
- Cycle **800 ADJUST XZ SYSTEM**
- Cycle**801 RESET ROTARY COORDINATE SYSTEM**
- Cycle **892 CHECK UNBALANCE**
- Cycle **1271 OCM RECTANGLE**
- Cycle **1272 OCM CIRCLE**
- Cycle **1273 OCM SLOT / RIDGE**
- Cycle **1274 OCM CIRCULAR SLOT**
- Cycle **1278 OCM POLYGON**
- Cycle **1281 OCM RECTANGLE BOUNDARY**
- Cycle **1282 OCM CIRCLE BOUNDARY**
- Cycles for coordinate transformation
- Cycles for grinding
- Touch probe cycles

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

Calling a cycle with CYCL CALL

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

Proceed as follows:

CYCL CALL

- Press the **CYCL CALL** key

CYCLE CALL M

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

Calling a cycle with CYCL CALL PAT

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

Further information: "Pattern definition with PATTERN DEF", Page 62

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

Calling a cycle with CYCL CALL POS

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

Using positioning logic, the control moves to the position defined in the **CYCL CALL POS** block:

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

> Programming and operating note:
> - Three coordinate axes must always be programmed in the **CYCL CALL POS** block. Using the coordinate in the tool axis, you can easily change the starting position. It serves as an additional datum shift.
> - The feed rate most recently defined in the **CYCL CALL POS** block is only used to traverse to the start position programmed in this block.
> - As a rule, the control moves without radius compensation (R0) to the position defined in the **CYCL CALL POS** block.
> - If you use **CYCL CALL POS** to call a cycle in which a start position is defined (e.g., Cycle **212**), then the position defined in the cycle serves as an additional shift of the position defined in the **CYCL CALL POS** block. You should therefore always define the start position in the cycle as 0.

Calling a cycle with M89/M99

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

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

To cancel the effect of **M89**:

- Program **M99** in the positioning block
- > The control moves to the last starting point.

  or
- Define a new machining cycle with **CYCL DEF**

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

Calling a cycle with SEL CYCLE

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

Proceed as follows:

PGM CALL
- Press the **PGM CALL** key

SELECT CYCLE
- Press the **SELECT CYCLE** soft key

SELECT FILE
- Press the **SELECT FILE** soft key
- Select NC program

Calling an NC program as a cycle

CYCL CALL
- Press the **CYCL CALL** key
- Press the soft key for the cycle call

  or
- Program **M99**

Programming and operating note:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
- When you execute an NC program selected with **SELECT CYCLE**, it will be executed in the Program Run, Single Block operating mode without stopping after each NC block. In addition, it is visible as a single NC block in the Program Run, Full Sequence operating mode.
- Please note that **CYCL CALL PAT** and **CYCL CALL POS** use a positioning logic before executing the cycle. With respect to the positioning logic, **SELECT CYCLE** and Cycle **12 PGM CALL** show the same behavior: In point pattern cycles, the clearance height is calculated based on the maximum value of all Z positions existing at the starting point of the pattern and all Z positions in the point pattern. With **CYCL CALL POS**, there will be no pre-positioning in the tool axis direction. This means that you need to manually program any pre-positioning in the file you call.

Working with a parallel axis

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

This is only possible when programming the following cycles:

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

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

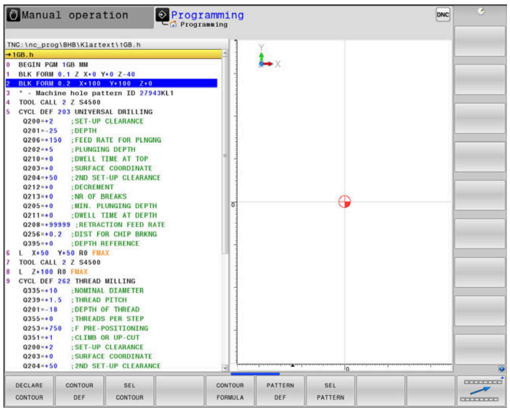
Further information: User's Manual for **Klartext Programming**

3.2 Program defaults for cycles

Overview

Some cycles always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** function you can define these cycle parameters at the beginning of the program, so that they are effective globally for all cycles used in the NC program. In the respective cycle you then simply reference the value defined at the beginning of the program.

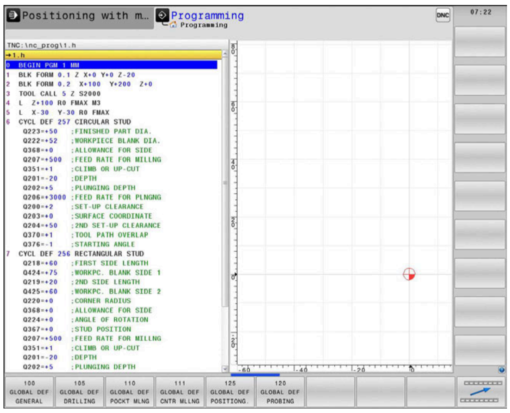
The following **GLOBAL DEF** functions are available:



Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	**GLOBAL DEF** GENERAL Definition of generally valid cycle parameters	57
105 GLOBAL DEF DRILLING	**GLOBAL DEF** DRILLING Definition of specific drilling cycle parameters	58
110 GLOBAL DEF POCKT MLNG	**GLOBAL DEF** POCKET MILLING Definition of specific pocket-milling cycle parameters	59
111 GLOBAL DEF CNTR MLLNG	**GLOBAL DEF** CONTOUR MILLING Definition of specific contour milling cycle parameters	60
125 GLOBAL DEF POSITIONG.	**GLOBAL DEF** POSITIONING Definition of the positioning behavior with **CYCL CALL PAT**	60
120 GLOBAL DEF PROBING	**GLOBAL DEF** PROBING Definition of specific touch probe cycle parameters	61

Entering GLOBAL DEF

Proceed as follows:



- Press the **Programming** key

SPEC FCT
- Press the **SPEC FCT** key

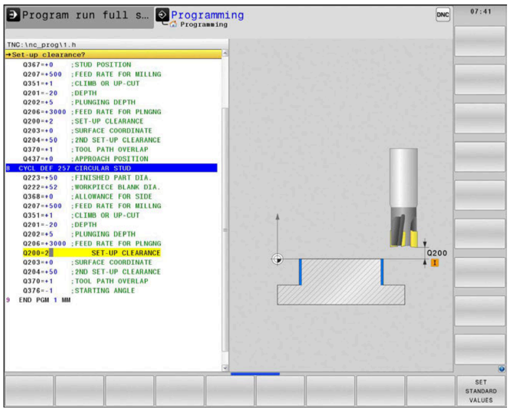
PROGRAM DEFAULTS
- Press the **PROGRAM DEFAULTS** soft key

GLOBAL DEF
- Press the **GLOBAL DEF** soft key

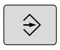
100 GLOBAL DEF GENERAL
- Select the desired **GLOBAL DEF** function (e.g., by pressing the **GLOBAL DEF GENERAL** soft key)
- Enter the required definitions
- Press the **ENT** key each time to confirm

Using GLOBAL DEF information

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



Proceed as follows:



- Press the **PROGRAMMING** key

- Press the **CYCL DEF** key

- Select the desired cycle group (e.g., pockets / studs / slot cycles)

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

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

NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly. There is a danger of collision!

- Make sure to use **GLOBAL DEF** carefully. Test your program before executing it
- If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

Global data valid everywhere

The parameters are valid for all **2xx** machining cycles as well as for Cycles **880, 1017**, **1018**, **1021**, **1022**, **1025** and touch probe cycles **451**, **452**, **453**

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
```

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
```

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
```

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
```

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
```

Global data for probing functions

The parameters apply to all touch-probe cycles **4xx** and **14xx** as well as the Cycles **271**, **286**, **287**, **880**, **1021**, **1022**, **1025**, **1271**, **1272**, **1273**, **1274**, **1278**

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

Example

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

3.3 Pattern definition with PATTERN DEF

Application

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

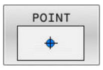
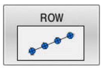
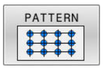
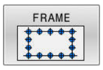
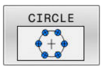
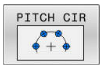
NOTICE

Danger of collision!

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

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

The following machining patterns are available:

Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	64
	ROW Definition of a single row, straight or rotated	65
	PATTERN Definition of a single pattern, straight, rotated or distorted	66
	FRAME Definition of a single frame, straight, rotated or distorted	68
	CIRCLE Definition of a full circle	70
	PITCH CIRCLE Definition of a pitch circle	71

Entering PATTERN DEF

Proceed as follows:

- Press the **PROGRAMMING** key
- Press the **SPEC FCT** key
- Press the **CONTOUR + POINT MACHINING** soft key
- Press the **PATTERN DEF** soft key
- Select the desired machining pattern (e.g., press the "single row" soft key)
- Enter the required definitions
- Press the **ENT** key each time to confirm

Using PATTERN DEF

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

Further information: "Calling a cycle", Page 49

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

Programming and operating note:

- A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
- The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the tool axis position from the cycle call or the value from cycle parameter **Q204** as the clearance height.
- If the coordinate surface in PATTERN DEF is larger than in the cycle, the set-up clearance and the 2nd set-up clearance reference the coordinate surface in PATTERN DEF.
- Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q345**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

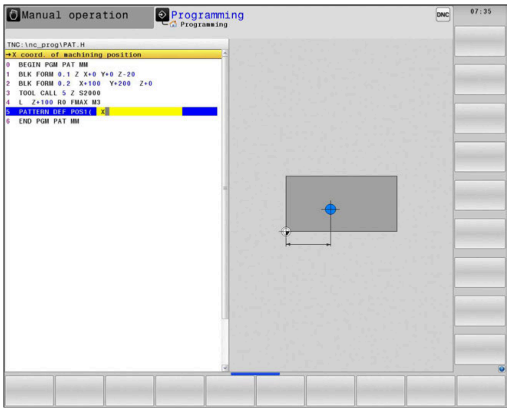
Operating note:

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

Defining individual machining positions

Programming and operating notes:

- You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.
- **POS1** must be programmed with absolute coordinates. **POS2** to **POS9** can be programmed as absolute or incremental values.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



Help graphic	Parameter
	POS1: **X coord. of machining position** Enter the X coordinate as an absolute value. Input: **-999999999...+999999999**
	POS1: **Y coord. of machining position** Enter the Y coordinate as an absolute value. Input: **-999999999...+999999999**
	POS1: **Coordinate of workpiece surface** Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**
	POS2: **X coord. of machining position** Enter the X coordinate as an incremental or absolute value. Input: **-999999999...+999999999**
	POS2: **Y coord. of machining position** Enter the Y coordinate as an incremental or absolute value. Input: **-999999999...+999999999**
	POS2: **Coordinate of workpiece surface** Enter the Z coordinate as an incremental or absolute value. Input: **-999999999...+999999999**

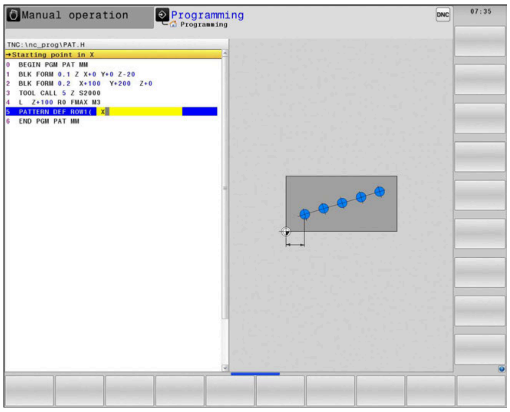
Example

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

Defining a single row

Programming and operating note:

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



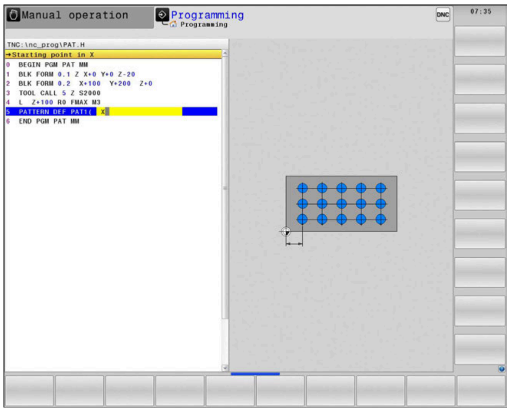
Help graphic	Parameter
	Starting point in X Coordinate of the starting point of the row in the X axis. This value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	Starting point in Y Coordinate of the starting point of the row in the Y axis. This value has an absolute effect. Input: **-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 )
```

Defining an individual pattern

> Programming and operating notes:
>
> - The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
> - If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



Help graphic	Parameter
	Starting point in X Absolute coordinate of the pattern starting point in the X axis Input: **-999999999...+999999999**
	Starting point in Y Absolute coordinate of the pattern starting point in the Y axis Input: **-999999999...+999999999**
	Spacing of machining positions X Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Spacing of machining positions Y Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Number of columns Total number of columns in the pattern Input: **0...999**
	Number of rows Total number of rows in the pattern Input: **0...999**
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**

Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**

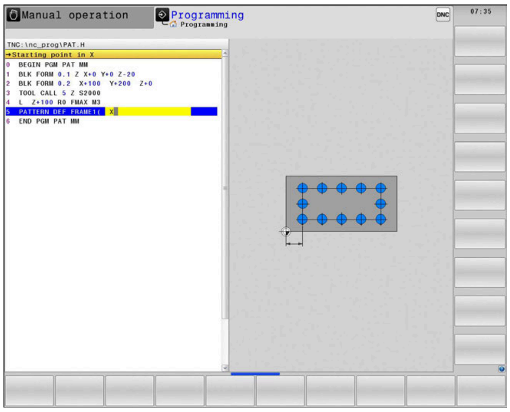
Example

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

Defining an individual frame

Programming and operating notes:

- The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



Help graphic	Parameter
	Starting point in X Absolute coordinate of the frame starting point in the X axis Input: **-999999999...+999999999**
	Starting point in Y Absolute coordinate of the frame starting point in the Y axis Input: **-999999999...+999999999**
	Spacing of machining positions X Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Spacing of machining positions Y Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Number of columns Total number of columns in the pattern Input: **0...999**
	Number of rows Total number of rows in the pattern Input: **0...999**
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: **-360.000...+360.000**

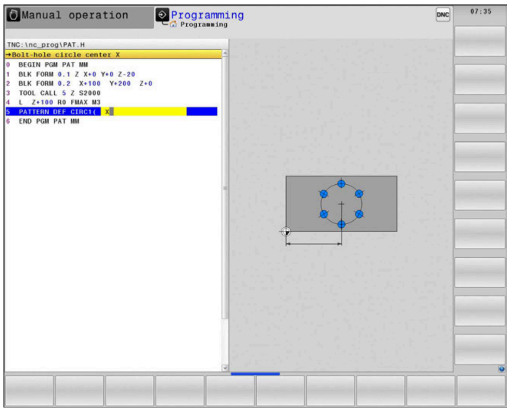
Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate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

Defining a full circle

> **i** Programming and operating notes:
> - If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



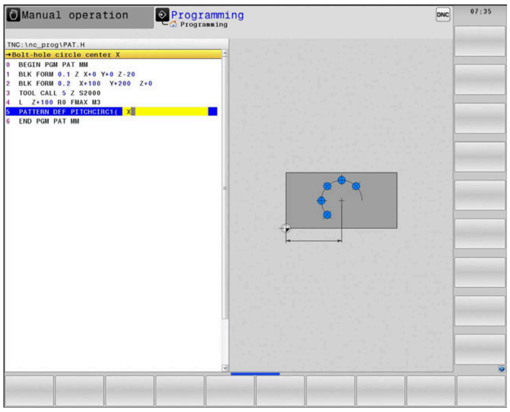
Help graphic	Parameter
	Bolt-hole circle center X Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter of the bolt hole circle Input: **0...999999999**
	Starting angle Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). You can enter a positive or negative value Input: **-360.000...+360.000**
	Number of operations Total number of machining positions on the circle Input: **0...999**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
CIRC1( X+25 Y+33 D80 START+45 NUM8 Z+0 )
```

Defining a pitch circle

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



Help graphic	Parameter
	Bolt-hole circle center X Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter of the bolt hole circle Input: **0...999999999**
	Starting angle Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). You can enter a positive or negative value Input: **-360.000...+360.000**
	Stepping angle/Stopping angle Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative you can enter the Stopping angle (switch via soft key) Input: **-360.000...+360.000**
	Number of operations Total number of machining positions on the circle Input: **0...999**
	Coordinate of workpiece surface Enter the Z coordinate at which machining starts. Input: **-999999999...+999999999**

Example

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

3.4 Point tables with cycles

Application with cycles

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

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

Related topics

- Contents of a point table, hiding individual points

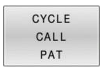
  Further information: User's Manual for **Klartext Programming**

Calling a cycle in connection with point tables

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

Proceed as follows:

- ▶ Press the **CYCL CALL** key



- ▶ Press the **CYCL CALL PAT** soft key
- ▶ Enter a feed rate

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

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

Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q345**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

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

Effect of the point table with SL cycles and Cycle 12

The control interprets the points as an additional datum shift.

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

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

Effect of the point table with Cycles 251 to 254

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

NOTICE

Danger of collision!

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

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

Programming and operating notes:

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

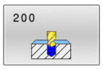
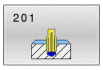
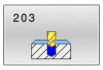
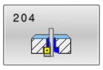
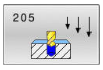
4

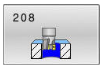
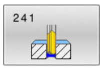
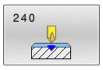
Cycles: Drilling

4.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	Cycle 200 DRILLING ■ Basic hole ■ Input of the dwell time at top and bottom ■ Depth reference selectable	78
	Cycle 201 REAMING ■ Reaming a hole ■ Input of the dwell time at bottom	82
	Cycle 202 REAMING ■ Boring a hole ■ Input of the retraction feed rate ■ Input of the dwell time at bottom ■ Input of the retracting movement	84
	Cycle 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	88
	Cycle 204 BACK BORING ■ Machining a counterbore on the underside of the workpiece ■ Input of the dwell time ■ Input of the retracting movement	94
	Cycle 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	98

Soft key	Cycle	Page
	Cycle 208 BORE MILLING ■ Milling of a hole ■ Input of a pre-drill diameter ■ Climb or up-cut milling selectable	106
	Cycle 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	111
	Cycle 240 CENTERING ■ Drilling a center hole ■ Input of the centering diameter or depth ■ Input of the dwell time at bottom	122

4.2 Cycle 200 DRILLING

ISO programming
G200

Application

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

Related topics

- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 88
- Cycle **205 UNIVERSAL PECKING** optionally with with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 98
- 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 111

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. The tool drills to the first plunging depth at the programmed feed rate **F**
3. The control retracts the tool at **FMAX** to set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to set-up clearance above the first plunging depth
4. The tool then drills deeper by the plunging depth at the programmed feed rate F.
5. The control repeats this procedure (steps 2 to 4) until the programmed depth is reached (the dwell time from **Q211** is effective with every infeed)
6. Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

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

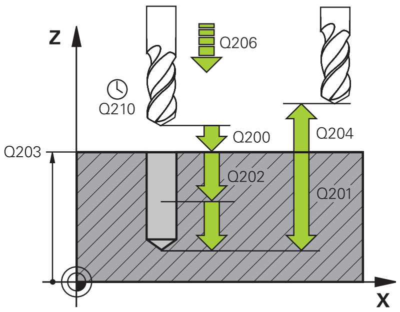
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**

Help graphic	Parameter
	Q395 Diameter as reference (0/1)? Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

Example

```
11 CYCL DEF 200 DRILLING ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q201=-20               ;DEPTH ~
    Q206=+150              ;FEED RATE FOR PLNGNG ~
    Q202=+5                ;PLUNGING DEPTH ~
    Q210=+0                ;DWELL TIME AT TOP ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q204=+50               ;2ND SET-UP CLEARANCE ~
    Q211=+0                ;DWELL TIME AT DEPTH ~
    Q395=+0                ;DEPTH REFERENCE
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
14 L  X+80  Y+50 FMAX M99
```

4.3 Cycle 201 REAMING

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

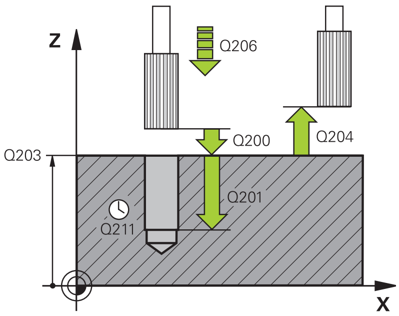
NOTICE
Danger of collision! If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision! ▶ Enter depth as negative ▶ 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 can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- 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
```

4.4 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 **Positioning w/ Manual Data Input** operating mode). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

NOTICE

Danger of collision!

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

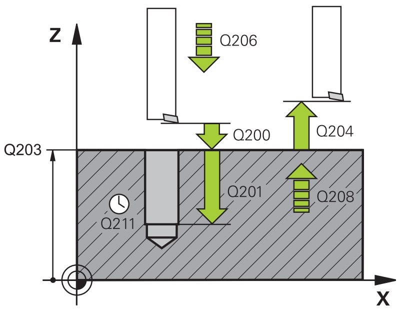
- Use **M137** to deactivate **M136** before the cycle start

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- If **Q214 DISENGAGING DIRECTN** is not 0, **Q357 CLEARANCE TO SIDE** is in effect.

Notes on programming

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

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while boring Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the feed rate for plunging applies. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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? 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? Distance between tool tooth and the wall. This value has an incremental effect. Only in effect if **Q214 DISENGAGING DIRECTN** is not 0. Input: **0...99999.9999**

Example

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

4.5 Cycle 203 UNIVERSAL DRILLING

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 78
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 98
- 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 111

Cycle run

Behavior without chip breaking, without decrement:

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**
4. Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
5. When machining without chip 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!

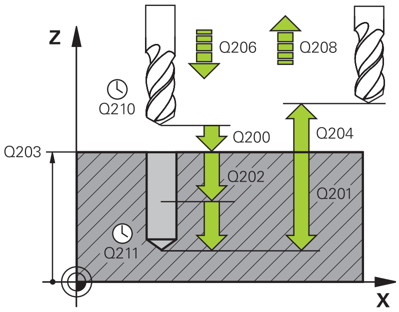
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

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

Help graphic	Parameter
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999** or **PREDEF**
	Q395 Diameter as reference (0/1)? Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

Example

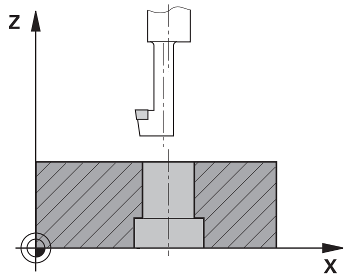
```
11 CYCL DEF 203 UNIVERSAL DRILLING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q201=-20            ;DEPTH ~
    Q206=+150           ;FEED RATE FOR PLNGNG ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q210=+0             ;DWELL TIME AT TOP ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q212=+0             ;DECREMENT ~
    Q213=+0             ;NR OF BREAKS ~
    Q205=+0             ;MIN. PLUNGING DEPTH ~
    Q211=+0             ;DWELL TIME AT DEPTH ~
    Q208=+99999         ;RETRACTION FEED RATE ~
    Q256=+0.2           ;DIST FOR CHIP BRKNG ~
    Q395=+0             ;DEPTH REFERENCE
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
```

4.6 Cycle 204 BACK BORING

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. Finally the tool moves at **FMAX** to set-up clearance.
7. The tool is again centered in the hole
8. The control restores the spindle status as it was at the cycle start.
9. If necessary, the control moves the tool to 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

NOTICE

Danger of collision!

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

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

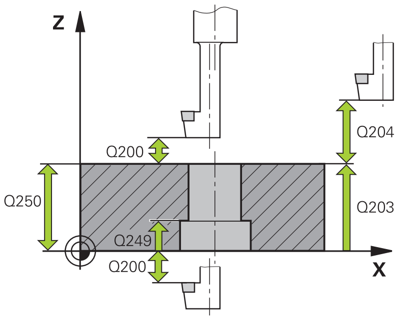
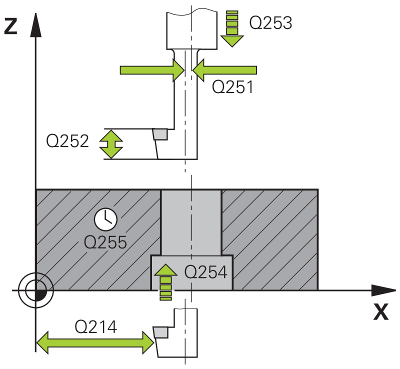
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.
- If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF COUNTERBORE Q249**, the control will display an error message.

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

Notes on programming

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

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q249 Depth of counterbore? Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q250 Material thickness? Height of the workpiece. Enter an incremental value. Input: **0.0001...99999.9999**
	**Q251 Tool edge off-center distance?** Off-center distance of the boring bar. Refer to the tool data sheet. This value has an incremental effect. Input: **0.0001...99999.9999**
	Q252 Tool edge height? Distance between underside of boring bar and main cutting tooth. Refer to the tool data sheet. This value has an incremental effect.
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q255 Dwell time in secs.? Dwell time in seconds at the bottom of the bore hole Input: **0...99999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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? 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=+5              ;DEPTH OF COUNTERBORE ~
    Q250=+20             ;MATERIAL THICKNESS ~
    Q251=+3.5            ;OFF-CENTER DISTANCE ~
    Q252=+15             ;TOOL EDGE HEIGHT ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q254=+200            ;F COUNTERBORING ~
    Q255=+0              ;DWELL TIME ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q214=+0              ;DISENGAGING DIRECTN ~
    Q336=+0              ;ANGLE OF SPINDLE
12 CYCL CALL
```

4.7 Cycle 205 UNIVERSAL PECKING

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 104

Related topics

- Cycle **200 DRILLING** for simple holes

  Further information: "Cycle 200 DRILLING", Page 78
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 88
- 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 111

Cycle run

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 deepened starting point in **Q379**, the control moves at the positioning feed rate **Q253 F PRE-POSITIONING** to the set-up clearance above the deepened starting point.
3 The tool drills at the programmed **Q206 FEED RATE FOR PLNGNG** to the plunging depth.
4 If you have programmed chip breaking, the control retracts the tool by the retraction value **Q256**.
5 Upon reaching the plunging depth, the control retracts the tool in the tool axis at the retraction feed rate **Q208** to the set-up clearance. The set-up clearance is above the **SURFACE COORDINATE Q203**.
6 The tool then moves at **Q373 FEED AFTER REMOVAL** to the entered advanced stop distance above the plunging depth last reached.
7 The tool drills at the feed in **Q206** to the next plunging depth. If a decrement Q212 is defined, the plunging depth is decreased after each infeed by the decrement.
8 The control repeats this procedure (steps 2 to 7) until the total drilling depth is reached.
9 If you entered a dwell time, the tool remains at the hole bottom for chip breaking. The control then retracts the tool at the retraction feed rate to the set-up clearance or the 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

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

Example:

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

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

Notes

NOTICE

Danger of collision!

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

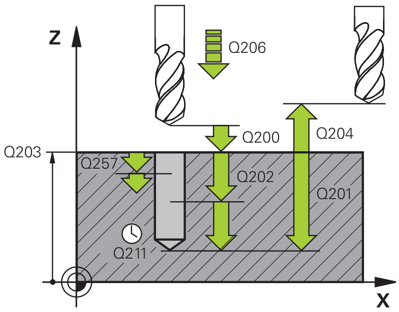
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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? 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 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? 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)? Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**
	Q373 Post-chip-removal approach feed? Traversing speed of the tool when approaching the advanced stop distance after chip removal. **0**: Move at **FMAX** **>0**: Feed in mm/min Input: **0...99999** or **FAUTO**, **FMAX**, **FU**, **FZ**

Example

```
11 CYCL DEF 205 UNIVERSAL PECKING ~
    Q200=+2            ;SET-UP CLEARANCE ~
    Q201=-20           ;DEPTH ~
    Q206=+150          ;FEED RATE FOR PLNGNG ~
    Q202=+5            ;PLUNGING DEPTH ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q204=+50           ;2ND SET-UP CLEARANCE ~
    Q212=+0            ;DECREMENT ~
    Q205=+0            ;MIN. PLUNGING DEPTH ~
    Q258=+0.2          ;UPPER ADV STOP DIST ~
    Q259=+0.2          ;LOWER ADV STOP DIST ~
    Q257=+0            ;DEPTH FOR CHIP BRKNG ~
    Q256=+0.2          ;DIST FOR CHIP BRKNG ~
    Q211=+0            ;DWELL TIME AT DEPTH ~
    Q379=+0            ;STARTING POINT ~
    Q253=+750          ;F PRE-POSITIONING ~
    Q208=+99999        ;RETRACTION FEED RATE ~
    Q395=+0            ;DEPTH REFERENCE ~
    Q373=+0            ;FEED AFTER REMOVAL
```

Chip removal and chip breaking

Chip removal

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

When the value entered in cycle parameter **Q202** is reached, the control performs chip removal. This means that the control always moves the tool to the retraction height, irrespective of the deepened starting point **Q379**. This height is calculated from **Q200 SET-UP CLEARANCE** + **Q203 SURFACE COORDINATE**

Example:

0 BEGIN PGM 205 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+0 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.2 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+10 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;DEPTH REFERENCE ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool
9 M30	; End of program
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
10 END PGM 205 MM	

4.8 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 run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance **Q200** above the workpiece surface
2. 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
- Press the **ENTER TEXT** soft key
- 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.

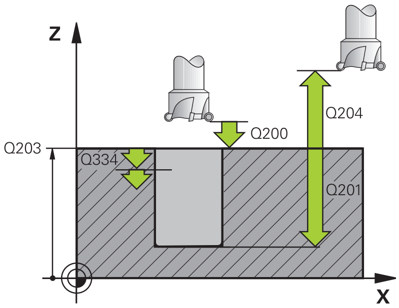
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have entered the bore hole diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation.
- An active mirror function **does not** influence the type of milling defined in the cycle.
- When calculating the overlap factor, the control takes the corner radius **DR2** of the current tool into account.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

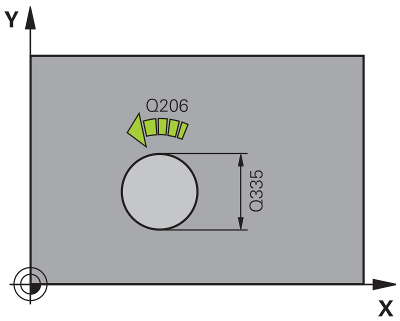
Notes on programming

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

Cycle parameters

Help graphic





Parameter

Q200 Set-up clearance?

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

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

Q201 Depth?

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

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

Q206 Feed rate for plunging?

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

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

Q334 Feed per revolution of helix

Depth of the tool plunge with each helix (=360°). 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 107

Input: **0...99999.9999**

Q342 Roughing diameter?

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 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q370 Path overlap factor? The control uses the path overlap factor to determine the stepover factor k. **0**: The control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly. **>0**: The control multiplies the factor by the active tool radius. The result is the stepover factor k. Input: **0.1...1999** or **PREDEF**

Example

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

4.9 Cycle 241 SINGLE-LIP D.H.DRLNG

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 78
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 88
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 98

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**
2 Depending on the positioning behavior, the control will either switch on the spindle with the programmed speed at the **SET-UP CLEARANCE Q200** or at a certain distance above the coordinate surface.

   Further information: "Position behavior when working with Q379", Page 118
3 The control executes the approach motion depending on the definition of **Q426 DIR. OF SPINDLE ROT.** with a spindle that rotates clockwise, counterclockwise, or is stationary
4 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**:
   - After defining **Q435 DWELL DEPTH**, the control reduces the feed rate by **Q401 FEED RATE FACTOR** after reaching the dwell depth and remains there for **Q211 DWELL TIME AT DEPTH**
   - If a smaller infeed value has been entered, the control drills to the plunging depth. The plunging depth is decreased after each infeed by **Q212 DECREMENT**
5 If programmed, the tool remains at the hole bottom for chip breaking.
6 After the control has reached the hole depth, 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 from **Q426**.
7 The control positions the tool to the retract position at **Q208 RETRACTION FEED RATE**.

   Further information: "Position behavior when working with Q379", Page 118
8 If programmed, the tool moves to 2nd set-up clearance at **FMAX**

Notes

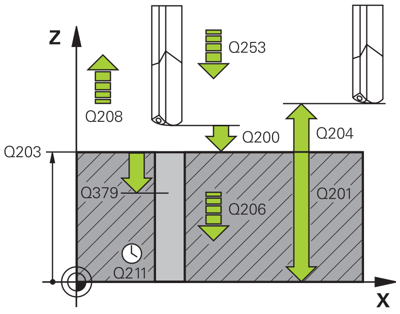
NOTICE
Danger of collision! If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision! ▶ Enter depth as negative ▶ 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

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

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and **Q203 SURFACE COORDINATE**. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between **Q203 SURFACE COORDINATE** and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. 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**
	Q379 Deepened starting point? If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the control retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input: **0...99999.999** or **FMAX**, **FAUTO**, **PREDEF**
	Q426 Rot. dir. of entry/exit (3/4/5)? Rotational speed at which the tool is to rotate when moving into and retracting from the hole. **3**: Spindle rotation with M3 **4**: Spindle rotation with M4 **5**: Movement with stationary spindle Input: **3**, **4**, **5**
	Q427 Spindle speed of entry/exit? Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input: **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 117 Input: **0...999**
	Q430 M function for coolant off? **>=0**: Miscellaneous function M for switching off the coolant. The control switches the coolant off if the tool is at **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 117 Input: **0...999**

Help graphic	Parameter
	Q435 Dwell depth? Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving the bottom of the hole in order to transport the chips to the top. Define a value smaller than **Q201 DEPTH**. This value has an incremental effect. Input: **0...99999.9999**
	Q401 Feed rate factor in %? Factor by which the control reduces the feed rate after reaching **Q435 DWELL DEPTH**. Input: **0.0001...100**
	Q202 Maximum plunging depth? Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect. Input: **0...99999.9999**
	Q212 Decrement? Value by which the control decreases **Q202 PLUNGING DEPTH** after each infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**

Example

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

User macro

A user macro is another NC program.

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

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

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

  Further information: Klartext Programming 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 V.ZEIT3	
9 LBL "Start"	
10 END PGM RET MM	

Position behavior when working 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.

Start of drilling

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

The starting point is at a certain value above the deepened starting point **Q379**. This value can be calculated as follows: 0.2 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 shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

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

Chip removal

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

The position for chip removal is calculated as follows:
0.8 x **Q379**=0.8*2=1.6; the position for chip removal is 1.6 mm 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 deepened starting point

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

4.10 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** feed rate to the deepened starting point.
4. The tool is centered at the programmed feed rate for plunging **F** to the programmed centering diameter or centering depth.
5. If a dwell time **Q211** is defined, the tool remains at the centering depth.
6. Finally, the tool is retracted to the set-up clearance or to the 2nd set-up clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

Notes

NOTICE

Danger of collision!

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

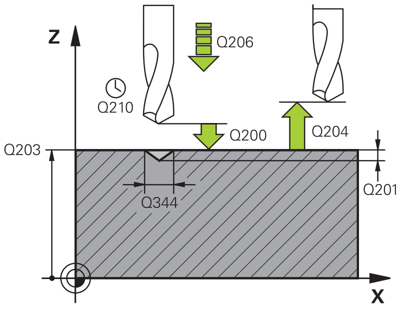
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- 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? **0**: There is no hole **>0**: Diameter of the pre-drilled hole Input: **0...99999.9999**

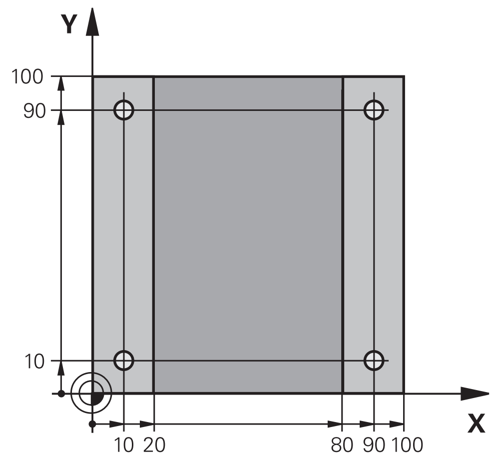
Help graphic	Parameter
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the deepened starting point. The speed is in mm/min. Only in effect if **Q342 ROUGHING DIAMETER** is not 0. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Example

```
11 CYCL DEF 240 CENTERING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q343=+1              ;SELECT DIA./DEPTH ~
    Q201=-2              ;DEPTH ~
    Q344=-10             ;DIAMETER ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q211=+0              ;DWELL TIME AT DEPTH ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q342=+12             ;ROUGHING DIAMETER ~
    Q253=+500            ;F PRE-POSITIONING
12 L  X+30  Y+20 R0 FMAX M3 M99
13 L  X+80  Y+50 R0 FMAX M99
```

4.11 Programming examples

Example: Drilling cycles



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

Example: Using cycles in 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: Tapping / Thread Milling", Page 129

0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	; Tool call: centering tool (tool radius 4)
4 L Z+50 R0 FMAX	; Move tool to clearance height
5 PATTERN DEF ~	
POS1(X+10 Y+10 Z+0) ~	
POS2(X+40 Y+30 Z+0) ~	
POS3(X+20 Y+55 Z+0) ~	
POS4(X+10 Y+90 Z+0) ~	
POS5(X+90 Y+90 Z+0) ~	
POS6(X+80 Y+65 Z+0) ~	
POS7(X+80 Y+30 Z+0) ~	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q343=+0 ;SELECT DIA./DEPTH ~	
Q201=-2 ;DEPTH ~	
Q344=-10 ;DIAMETER ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q342=+0 ;ROUGHING DIAMETER ~	
Q253=+750 ;F PRE-POSITIONING	
7 GLOBAL DEF 125 POSITIONING ~	
Q345=+1 ;SELECT POS. HEIGHT	
8 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
9 L Z+100 R0 FMAX	; Retract the tool
10 TOOL CALL 227 Z S5000	; Tool call: drill (radius 2.4)

11 L X+50 R0 F5000	; Move tool to clearance height
12 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
13 CYCL CALL PAT F500 M3	; Cycle call in connection with the point pattern
14 L Z+100 R0 FMAX	; Retract the tool
15 TOOL CALL 263 Z S200	; Tool call: tap (radius 3)
16 L Z+100 R0 FMAX	; Move tool to clearance height
17 CYCL DEF 206 TAPPING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH OF THREAD ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE	
18 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
19 L Z+100 R0 FMAX	; Retract the tool
20 M30	; End of program
21 END PGM 1 MM	

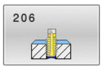
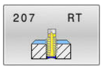
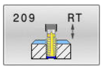
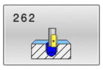
5

Cycles: Tapping / Thread Milling

5.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	Cycle 206 TAPPING ■ With a floating tap holder ■ Input of the dwell time at bottom	131
	Cycle 207 RIGID TAPPING ■ Without a floating tap holder ■ Input of the dwell time at bottom	134
	Cycle 209 TAPPING W/ CHIP BRKG ■ Without a floating tap holder ■ Input of chip breaking behavior	139
	Cycle 262 THREAD MILLING ■ Milling a thread into pre-drilled material	146
	Cycle 263 THREAD MLLNG/CNTSNKG ■ Milling a thread into pre-drilled material ■ Machining a countersunk chamfer	151
	Cycle 264 THREAD DRILLNG/MLLNG ■ Drilling into solid material ■ Milling a thread	157
	Cycle 265 HEL. THREAD DRLG/MLG ■ Milling a thread into solid material	163
	Cycle 267 OUTSIDE THREAD MLLNG ■ Milling an external thread ■ Machining a countersunk chamfer	168

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 134
- Cycle **209 TAPPING W/ CHIP BRKG** without floating tap holder, but optionally with chip breaking
  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 139

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to set-up clearance at the end of the dwell time. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
4. At the set-up clearance, the direction of spindle rotation reverses once again.

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

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.
- In Cycle **206**, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

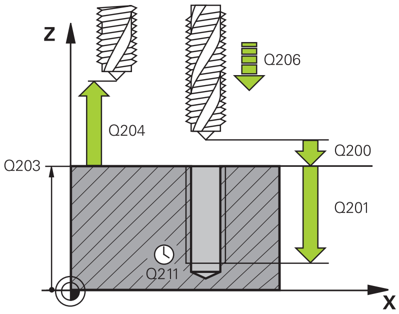
Notes on programming

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

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603):
    FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
    SpindlePotentiometer (feed rate override is not active)
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Guide value: 4 times the thread pitch Input: **0...99999.9999** or **PREDEF**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool during tapping Input: **0...99999.999** or **FAUTO**
	Q211 Dwell time at the depth? Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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)

Retracting after a program interruption

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

5.3 Cycle 207 RIGID TAPPING

ISO programming
G207

Application

> Refer to your machine manual.
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
> This cycle is effective only for machines with servo-controlled spindle.

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

Related topics

- Cycle **206 TAPPING** with floating tap holder
  Further information: "Cycle 206 TAPPING ", Page 131
- Cycle **209 TAPPING W/ CHIP BRKG** without floating tap holder, but optionally with chip breaking
  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 139

Cycle run

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

> 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **M3** (or **M4**) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the **TOOL CALL** block).
- If you do not program **M3** (or **M4**) before this cycle, the spindle will stand still after the end of the cycle. In this case, you must restart the spindle with **M3** (or **M4**) before the next operation.
- If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

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

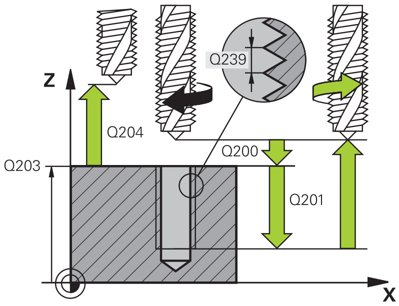
Notes on programming

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

Note regarding machine parameters

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

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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

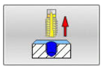
```
11 CYCL DEF 207 RIGID TAPPING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q201=-18            ;DEPTH OF THREAD ~
    Q239=+1             ;THREAD PITCH ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

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



- Press the retract soft key.

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

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

Proceed as follows:

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

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

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

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

NOTICE

Danger of collision!

If you retract the tool manually and move it in the negative direction instead of the positive direction, there is a danger of collision.

- With a manual retraction you can move the tool in the positive as well as the negative direction of the tool axis.
- Before starting the manual retraction, you should make yourself fully aware of the direction into which you move the tool out of the hole.

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 131
- Cycle **207 RIGID TAPPING** without floating tap holder
  Further information: "Cycle 207 RIGID TAPPING ", Page 134

Cycle run

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

> 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **M3** (or **M4**) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the **TOOL CALL** block).
- If you do not program **M3** (or **M4**) before this cycle, the spindle will stand still after the end of the cycle. In this case, you must restart the spindle with **M3** (or **M4**) before the next operation.
- If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

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

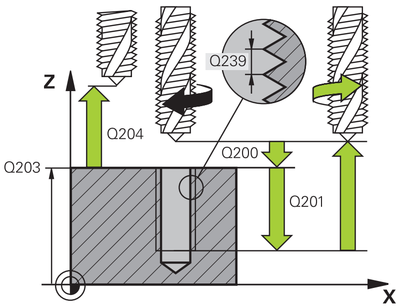
Notes on programming

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

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603):
    FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
    SpindlePotentiometer (feed rate override is not active)
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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**

Help graphic	Parameter
	Q336 Angle for spindle orientation? Angle to which the control positions the tool before machining the thread. This allows you to re-cut the thread, if required. This value has an absolute effect. Input: **0...360**
	Q403 RPM factor for retraction? Factor by which the control increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole. Maximum increase to maximum speed of the active gear stage. Input: **0.0001...10**

Example

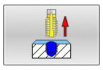
```
11 CYCL DEF 209 TAPPING W/ CHIP BRKG ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q201=-18             ;DEPTH OF THREAD ~
    Q239=+1              ;THREAD PITCH ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q257=+0              ;DEPTH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

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

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



- Press the retract soft key.

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

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

Proceed as follows:

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

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

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

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

NOTICE

Danger of collision!

If you retract the tool manually and move it in the negative direction instead of the positive direction, there is a danger of collision.

- With a manual retraction you can move the tool in the positive as well as the negative direction of the tool axis.
- Before starting the manual retraction, you should make yourself fully aware of the direction into which you move the tool out of the hole.

5.5 Fundamentals of thread milling

Requirements

- Your machine tool features internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars)
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer (you can set the compensation in **TOOL CALL** using the **DR** delta radius).
- If you are using a left-cutting tool (**M4**), the type of milling in **Q351** is reversed
- The working direction is determined by the following input parameters: Algebraic sign **Q239** (+ = right-hand thread / – = left-hand thread) and type of milling **Q351** (+1 = climb / –1 = up-cut).

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

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

Programming and operating notes:

- The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle **8 MIRRORING** in only one axis.
- The programmed feed rate for thread milling references the cutting edge of the tool. However, since the control always displays the feed rate relative to the center path of the tool tip, the displayed value does not match the programmed value.

5.6 Cycle 262 THREAD MILLING

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 151
- 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 157
- 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 163
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 168

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3 The tool then approaches the nominal thread diameter tangentially in a helical movement. Before the helical approach, a compensating movement of the tool axis is carried out in order to begin at the programmed starting plane for the thread path
4 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
5 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
6 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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

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!

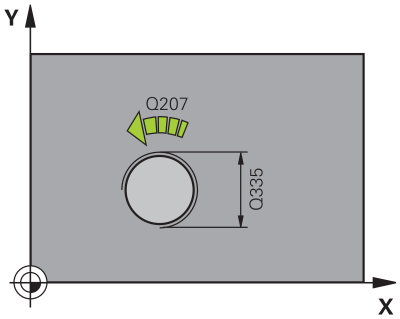
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.

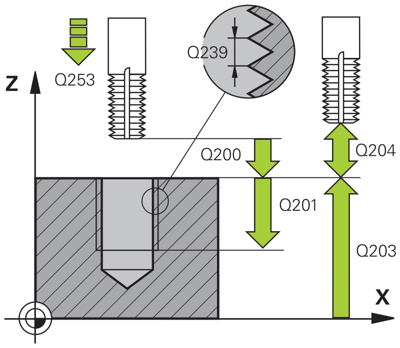
Notes on programming

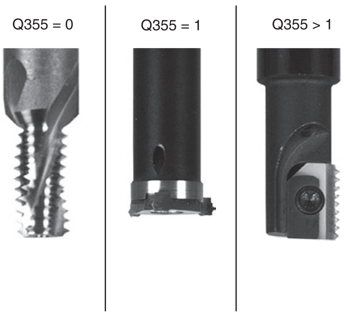
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If you program the thread depth =0, the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

Cycle parameters

Help graphic







Parameter

Q335 Nominal diameter?

Nominal thread diameter

Input: **0...99999.9999**

Q239 Pitch?

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

+= right-hand thread

– = left-hand thread

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

Q201 Depth of thread?

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

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

Q355 Number of threads per step?

Number of thread revolutions by which the tool is moved:

0 = one helical line to the thread depth

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

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

Input: **0...99999**

Q253 Feed rate for pre-positioning?

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

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

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

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

+1 = climb milling

–1 = up-cut milling

(if you enter 0, climb milling is performed)

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

Q200 Set-up clearance?

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

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

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. 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? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 262 THREAD MILLING ~
    Q335=+5              ;NOMINAL DIAMETER ~
    Q239=+1              ;THREAD PITCH ~
    Q201=-18             ;DEPTH OF THREAD ~
    Q355=+0              ;THREADS PER STEP ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q351=+1              ;CLIMB OR UP-CUT ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q207=+500            ;FEED RATE MILLING ~
    Q512=+0              ;FEED FOR APPROACH
12 CYCL CALL
```

5.7 Cycle 263 THREAD MLLNG/CNTSNKG

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 146
- 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 157
- 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 163
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 168

Cycle run

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

Countersinking

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

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

4 Then, depending on the available space, the control smoothly approaches the tool to the core diameter, either tangentially from the center or with a pre-positioning movement to the side, and follows a circular path

Countersinking at front

5 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

6 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking

7 The tool then moves in a semicircle to the hole center

Thread milling

8 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut)

9 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion

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

11 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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

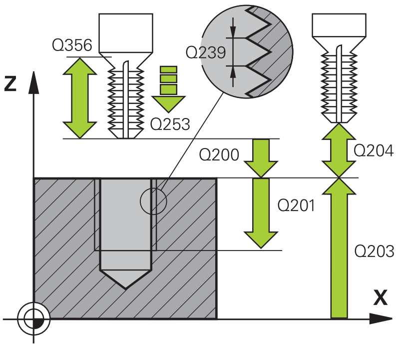
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1 Depth of thread
  2 Countersinking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

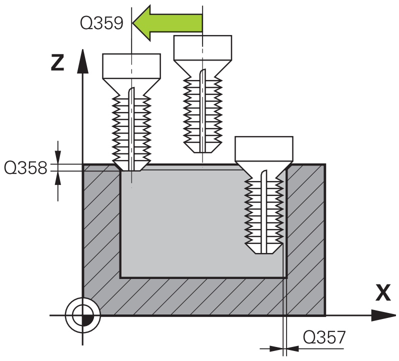
Notes on programming

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

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

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q356 Countersinking depth? Distance between tool point and the top surface of the workpiece. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	**Q357 Safety clearance to the side?** Distance between tool tooth and the wall. This value has an incremental effect. Input: **0...99999.9999**
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 263 THREAD MLLNG/CNTSNKG ~
    Q335=+5                  ;NOMINAL DIAMETER ~
    Q239=+1                  ;THREAD PITCH ~
    Q201=-18                 ;DEPTH OF THREAD ~
    Q356=-20                 ;COUNTERSINKING DEPTH ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q351=+1                  ;CLIMB OR UP-CUT ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q357=+0.2                ;CLEARANCE TO SIDE ~
    Q358=+0                  ;DEPTH AT FRONT ~
    Q359=+0                  ;OFFSET AT FRONT ~
    Q203=+0                  ;SURFACE COORDINATE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q254=+200                ;F COUNTERBORING ~
    Q207=+500                ;FEED RATE MILLING ~
    Q512=+0                  ;FEED FOR APPROACH
12 CYCL CALL
```

5.8 Cycle 264 THREAD DRILLNG/MLLNG

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 146
- 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 151
- 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 163
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 168

Cycle run

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

Drilling

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

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

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

5 The control repeats this procedure (steps 2 to 4) until the total drilling depth is reached

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

7 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking

8 The tool then moves in a semicircle to the hole center

Thread milling

9 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut)

10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion

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

12 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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

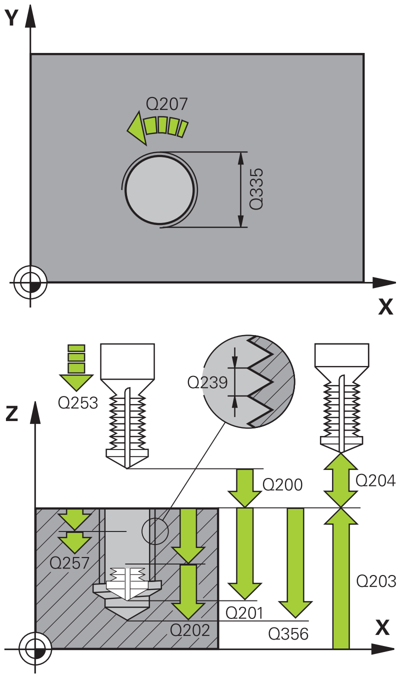
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1 Depth of thread
  2 Countersinking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

Notes on programming

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

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

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q356 Total hole depth? Distance between workpiece surface and hole bottom. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q202 Maximum plunging depth? Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect. The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if: ■ the plunging depth is equal to the depth ■ the plunging depth is greater than the depth Input: **0...99999.9999**
	Q258 Upper advanced stop distance? Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after first chip removal. This value has an incremental effect. Input: **0...99999.9999**

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

5.9 Cycle 265 HEL. THREAD DRLG/MLG

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 146
- 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 151
- 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 157
- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 168

Cycle run

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

Countersinking at front

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

Thread milling

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

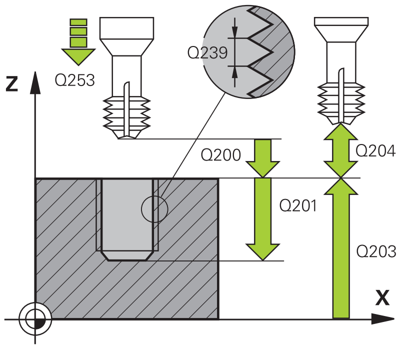
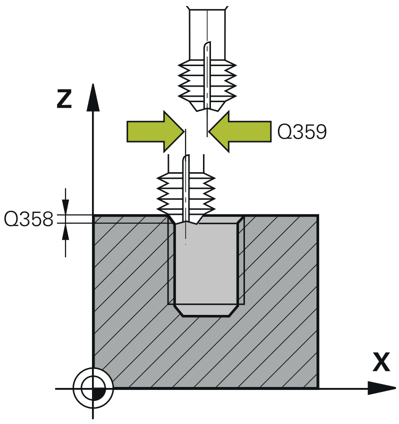
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

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

Notes on programming

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

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	**Q358 Sinking depth at front?** Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q360 Countersink (before/after:0/1)? Execution of the chamfer **0** = before thread machining **1** = after thread machining Input: **0**, **1**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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 PROCESS ~
    Q200=+2          ;SET-UP CLEARANCE ~
    Q203=+0          ;SURFACE COORDINATE ~
    Q204=+50         ;2ND SET-UP CLEARANCE ~
    Q254=+200        ;F COUNTERBORING ~
    Q207=+500        ;FEED RATE MILLING
12 CYCL CALL
```

5.10 Cycle 267 OUTSIDE THREAD MLLNG

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 146
- 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 151
- 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 157
- 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 163

Cycle run

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

Countersinking at front

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

Thread milling

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

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!

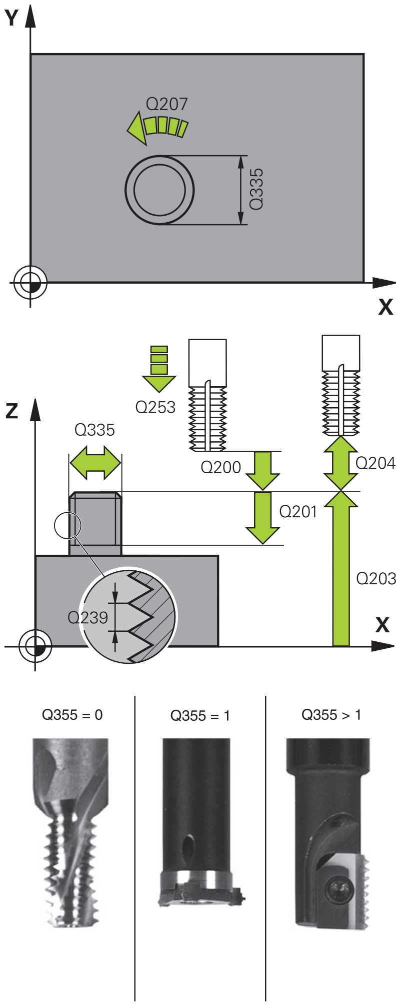
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

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

Notes on programming

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

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q355 Number of threads per step? Number of thread revolutions by which the tool is moved: **0** = one helical line to the thread depth **1** = continuous helical path over the entire length of the thread **>1** = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch. Input: **0...99999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. 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? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

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

5.11 Programming examples

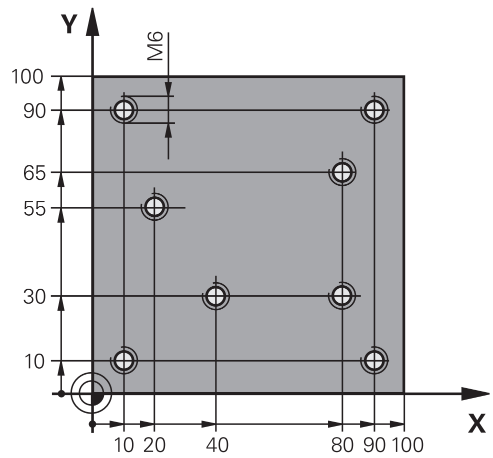
Example: Thread milling

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

The tool radii have been selected in such a way that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping



0 BEGIN PGM TAP MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	; Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 171 Z S5000	; Tool call: centering tool
4 L Z+100 R0 FMAX M3	; Move tool to clearance height (program a value for F): the control positions the tool at the clearance height after every cycle
5 CYCL DEF 240 CENTERING ~	; Cycle definition: Centering
Q200=+2 ;SET-UP CLEARANCE ~	
Q343=+1 ;SELECT DIA./DEPTH ~	
Q201=-1 ;DEPTH ~	
Q344=-7 ;DIAMETER ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE	
6 CALL LBL 1	
7 L Z+100 R0 FMAX	; Retract the tool
8 TOOL CALL 227 Z S5000	; Tool call: drill
9 L Z+100 R0 FMAX M3	; Move tool to clearance height (enter a value for F)
10 CYCL DEF 200 DRILLING ~	; Cycle definition: Drilling
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	

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

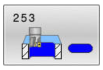
6

Cycles: Pocket Milling / Stud Milling / Slot Milling

6.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	Cycle 251 RECTANGULAR POCKET ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	179
	Cycle 252 CIRCULAR POCKET ■ Roughing and finishing cycle ■ Plunging strategy: helical or vertical	187
	Cycle 253 SLOT MILLING ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	194
	Cycle 254 CIRCULAR SLOT ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	201
	Cycle 256 RECTANGULAR STUD ■ Roughing and finishing cycle ■ Approach position: selectable	208
	Cycle 257 CIRCULAR STUD ■ Roughing and finishing cycle ■ Input of the start angle ■ Helical infeed starting from the workpiece blank diameter	214
	Cycle 258 POLYGON STUD ■ Roughing and finishing cycle ■ Helical infeed starting from the workpiece blank diameter	219
	Cycle 233 FACE MILLING ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls	225

6.2 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

5 If finishing allowances have been defined, the control plunges and then approaches the contour. The approach movement occurs on a radius in order to ensure a gentle approach. The control first finishes the pocket walls, with multiple infeeds, if so specified.
6 Then the control finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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 186

Notes on programming

- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

Cycle parameters

Help graphic | **Parameter**

Q215 Machining operation (0/1/2)?

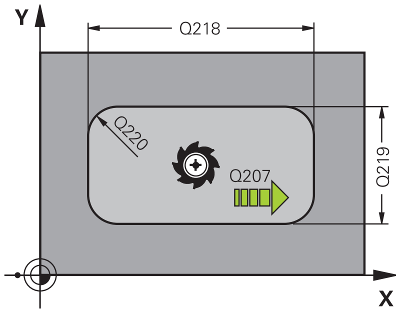
Define the machining operation:

0: Roughing and finishing

1: Only roughing

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

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



Q218 First side length?

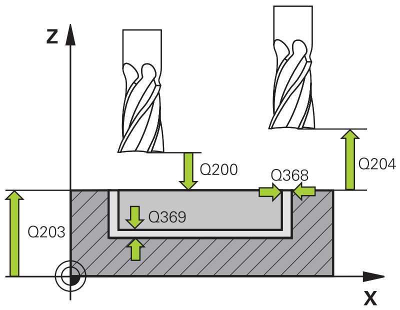
Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q219 Second side length?

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

Input: **0...99999.9999**



Q220 Corner radius?

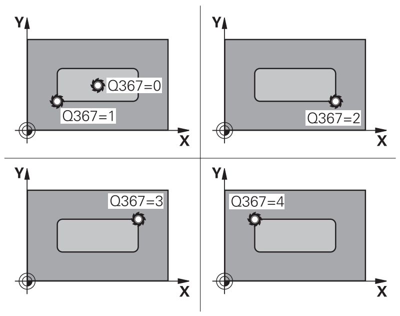
Radius of the pocket corner. If you have entered 0 here, the control assumes that the corner radius is equal to the tool radius.

Input: **0...99999.9999**

Q368 Finishing allowance for side?

Finishing allowance in the 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

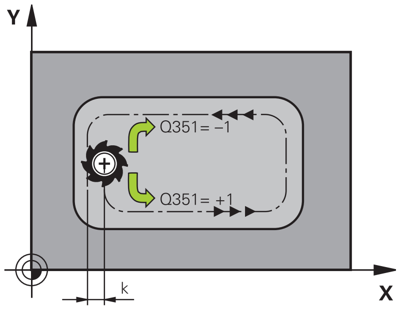
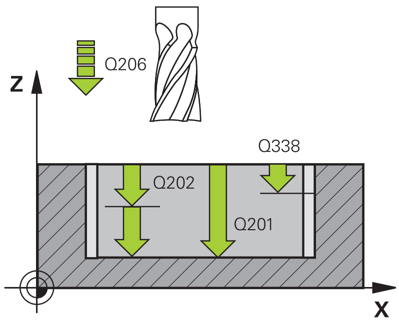
4: Tool position = Upper left corner

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**

Help graphic	Parameter
	**Q351 Direction? Climb=+1, Up-cut=-1** Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and bottom of pocket. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance 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 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. 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 186
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

11 CYCL DEF 251 RECTANGULAR POCKET ~	
Q215=+0	;MACHINING OPERATION ~
Q218=+60	;FIRST SIDE LENGTH ~
Q219=+20	;2ND SIDE LENGTH ~
Q220=+0	;CORNER RADIUS ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q224=+0	;ANGLE OF ROTATION ~
Q367=+0	;POCKET POSITION ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-20	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q338=+0	;INFEED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q370=+1	;TOOL PATH OVERLAP ~
Q366=+1	;PLUNGE ~
Q385=+500	;FINISHING FEED RATE ~
Q439=+0	;FEED RATE REFERENCE
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.

6.3 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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 193

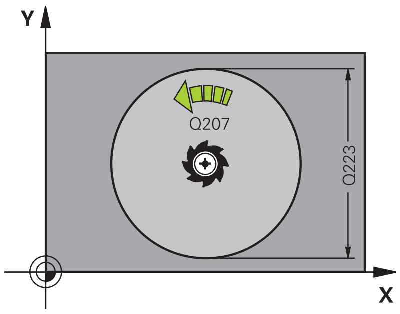
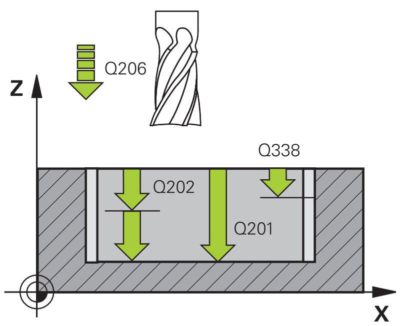
Notes on programming

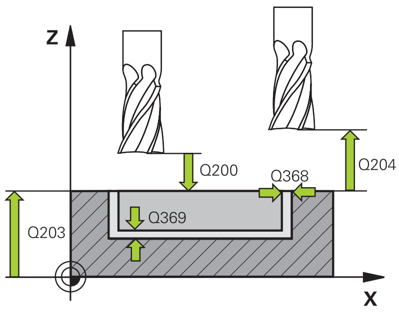
- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position (circle center) with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines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

Note regarding machine parameters

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

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q223 Circle diameter?** Diameter of the finished pocket Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the 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**

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**
	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. 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 193

Help graphic	Parameter
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 252 CIRCULAR POCKET ~
    Q215=+0              ;MACHINING OPERATION ~
    Q223=+50             ;CIRCLE DIAMETER ~
    Q368=+0              ;ALLOWANCE FOR SIDE ~
    Q207=+500            ;FEED RATE MILLING ~
    Q351=+1              ;CLIMB OR UP-CUT ~
    Q201=-20             ;DEPTH ~
    Q202=+5              ;PLUNGING DEPTH ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q338=+0              ;INFEED FOR FINISHING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q370=+1              ;TOOL PATH OVERLAP ~
    Q366=+1              ;PLUNGE ~
    Q385=+500            ;FINISHING FEED RATE ~
    Q439=+0              ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

Plunging strategy Q366 with RCUTS

Behavior with RCUTS

Helical plunging **Q366=1**:

RCUTS > 0

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

  $Helicalradius = R_{corr} - RCUTS$

  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.

6.4 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

5. If a finishing allowance has been defined during pre-machining, the control first finishes the slot walls, using multiple infeeds, if so specified. The slot wall is approached tangentially in the left slot arc
6. Then the control finishes the floor of the slot from the inside out.

Notes

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start! 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!

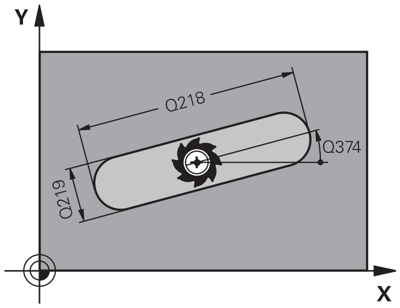
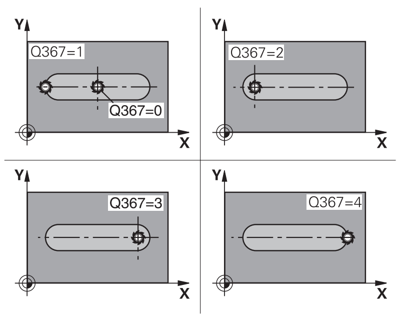
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

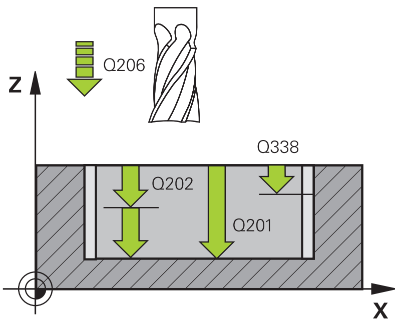
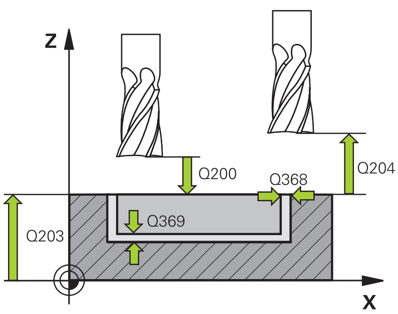
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 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 plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q218 Length of slot?** Enter the length of the slot. It is parallel to the main axis of the working plane. 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. 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**
	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**

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? 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. Alternative: **PREDEF** Input: **0**, **1**, **2**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 253 SLOT MILLING ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q218=+60                ;SLOT LENGTH ~
    Q219=+10                ;SLOT WIDTH ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q374=+0                 ;ANGLE OF ROTATION ~
    Q367=+0                 ;SLOT POSITION ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;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=+3                 ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

6.5 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

5 If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially
6 Then the control finishes the floor of the slot from the inside out

Notes

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start! 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.

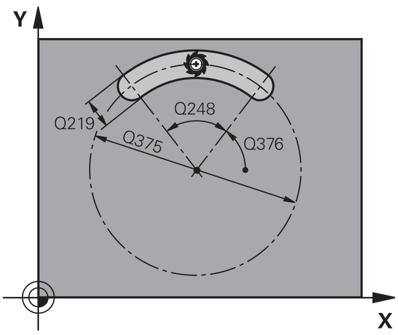
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

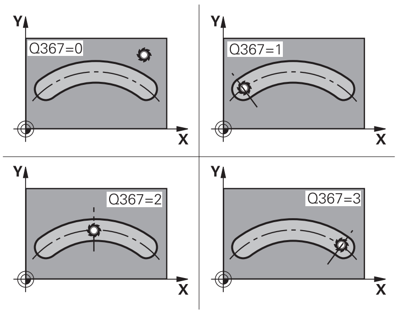
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 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.

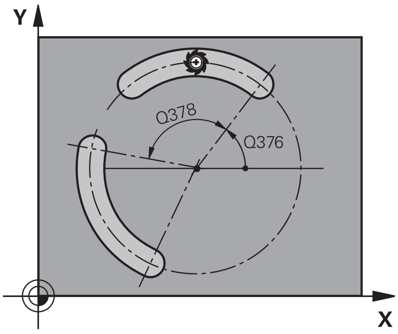
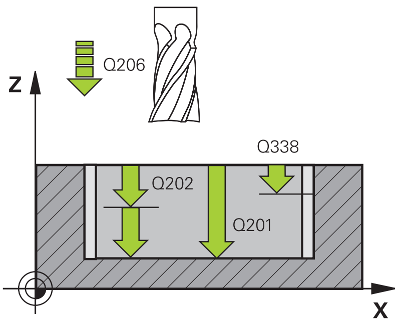
Notes on programming

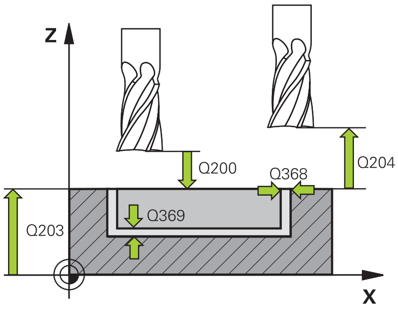
- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
 	**Q219 Width of slot?** Enter the width of the slot, 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. 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**
	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**

Help graphic	Parameter
	**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**
	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**

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**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The plunging angle **ANGLE** in the tool table is not evaluated. **1, 2**: Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message **PREDEF**: The control uses the value from the GLOBAL DEF block. Input: **0**, **1**, **2**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0            ;MACHINING OPERATION ~
    Q219=+10           ;SLOT WIDTH ~
    Q368=+0            ;ALLOWANCE FOR SIDE ~
    Q375=+60           ;PITCH CIRCLE DIAMETR ~
    Q367=+0            ;REF. SLOT POSITION ~
    Q216=+50           ;CENTER IN 1ST AXIS ~
    Q217=+50           ;CENTER IN 2ND AXIS ~
    Q376=+0            ;STARTING ANGLE ~
    Q248=+0            ;ANGULAR LENGTH ~
    Q378=+0            ;STEPPING ANGLE ~
    Q377=+1            ;NR OF REPETITIONS ~
    Q207=+500          ;FEED RATE MILLING ~
    Q351=+1            ;CLIMB OR UP-CUT ~
    Q201=-20           ;DEPTH ~
    Q202=+5            ;PLUNGING DEPTH ~
    Q369=+0            ;ALLOWANCE FOR FLOOR ~
    Q206=+150          ;FEED RATE FOR PLNGNG ~
    Q338=+0            ;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
12 L  X+50  Y+50 R0 FMAX M99
```

6.6 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.

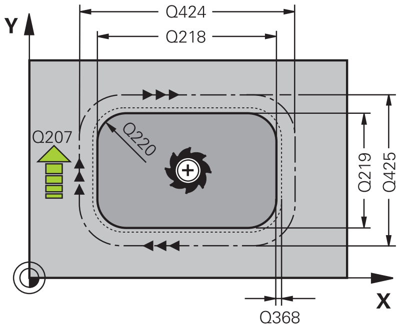
- Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed.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

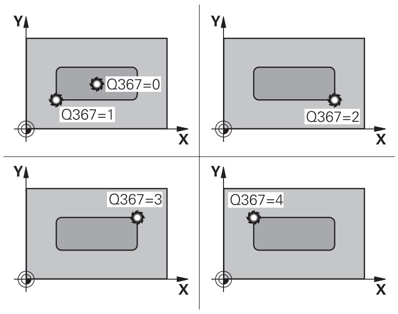
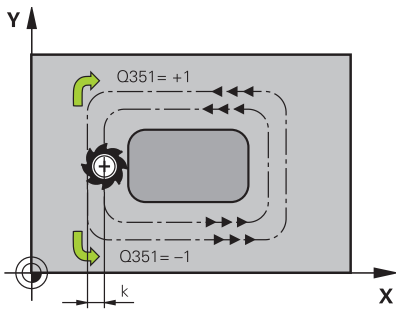
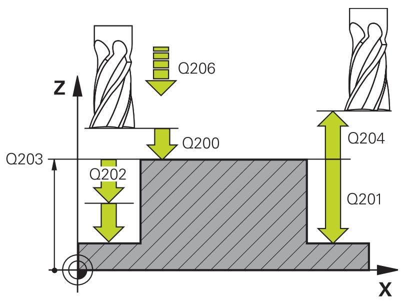
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 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)? 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)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	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 256 RECTANGULAR STUD ~
    Q218=+60            ;FIRST SIDE LENGTH ~
    Q424=+75            ;WORKPC. BLANK SIDE 1 ~
    Q219=+20            ;2ND SIDE LENGTH ~
    Q425=+60            ;WORKPC. BLANK SIDE 2 ~
    Q220=+0             ;CORNER RADIUS ~
    Q368=+0             ;ALLOWANCE FOR SIDE ~
    Q224=+0             ;ANGLE OF ROTATION ~
    Q367=+0             ;STUD POSITION ~
    Q207=+500           ;FEED RATE MILLING ~
    Q351=+1             ;CLIMB OR UP-CUT ~
    Q201=-20            ;DEPTH ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q206=+3000          ;FEED RATE FOR PLNGNG ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q370=+1             ;TOOL PATH OVERLAP ~
    Q437=+0             ;APPROACH POSITION ~
    Q215=+1             ;MACHINING OPERATION ~
    Q369=+0             ;ALLOWANCE FOR FLOOR ~
    Q338=+0             ;INFEED FOR FINISHING ~
    Q385=+500           ;FEED RATE FOR FINISHING
12 L  X+50  Y+50 R0 FMAX M99
```

6.7 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 firsts departs on a tangential path and is then retracted in the tool axis to the 2nd set-up clearance defined in the cycle. This means that the end position differs from the starting position

Notes

NOTICE

Danger of collision!

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

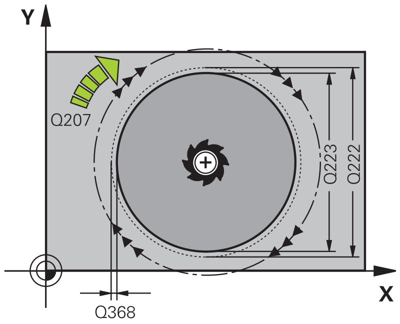
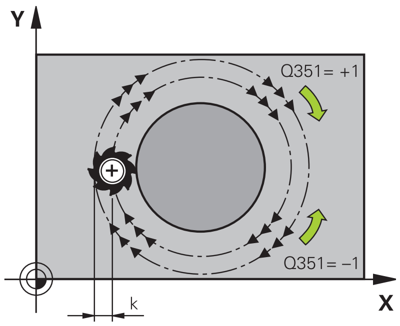
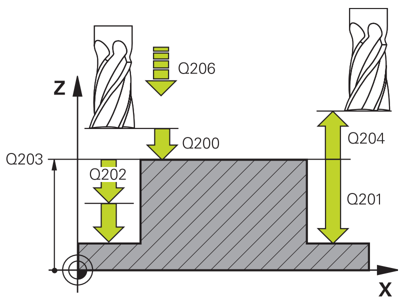
- Check the machining sequence using the graphic simulation.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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? Polar angle relative to the stud center, from which the tool approaches the stud. Input: **-1...+359**
	Q215 Machining operation (0/1/2)? Specify the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? Finishing allowance 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 257 CIRCULAR STUD ~
    Q223=+50            ;FINISHED PART DIA. ~
    Q222=+52            ;WORKPIECE BLANK DIA. ~
    Q368=+0             ;ALLOWANCE FOR SIDE ~
    Q207=+500           ;FEED RATE MILLING ~
    Q351=+1             ;CLIMB OR UP-CUT ~
    Q201=-20            ;DEPTH ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q206=+3000          ;FEED RATE FOR PLNGNG ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q370=+1             ;TOOL PATH OVERLAP ~
    Q376=-1             ;STARTING ANGLE ~
    Q215=+1             ;MACHINING OPERATION ~
    Q369=+0             ;ALLOWANCE FOR FLOOR ~
    Q338=+0             ;INFEED FOR FINISHING ~
    Q385=+500           ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

6.8 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, check the end position of the tool after the cycle
- After the cycle, program absolute coordinates (no incremental coordinates)

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

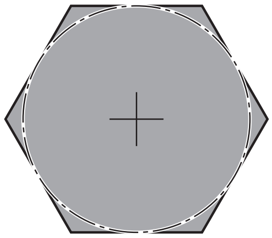
Notes on programming

- 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.

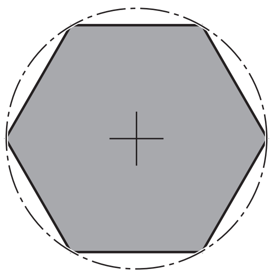
Cycle parameters

Help graphic

Q573 = 0



Q573 = 1



Parameter

Q573 Inscr.circle/circumcircle (0/1)?

Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle:

0: Dimension is referenced to the inscribed circle

1: Dimension is referenced to the circumcircle

Input: **0**, **1**

Q571 Reference circle diameter?

Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed.

Input: **0...99999.9999**

Q222 Workpiece blank diameter?

Enter the diameter of the blank. The workpiece blank diameter must be greater than the reference circle diameter. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover.

Input: **0...99999.9999**

Q572 Number of corners?

Enter the number of corners of the polygon stud. The control distributes the corners evenly on the stud.

Input: **3...30**

Q224 Angle of rotation?

Specify which angle is used to machine the first corner of the polygon stud.

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

Q220 Radius / Chamfer (+/-)?

Enter the value for the radius or chamfer form element. If you enter a positive value, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer.

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

Q368 Finishing allowance for side?

Finishing allowance in the working plane. If you enter a negative value here, the control will return the tool to a diameter outside of the workpiece blank diameter after roughing. This value has an incremental effect.

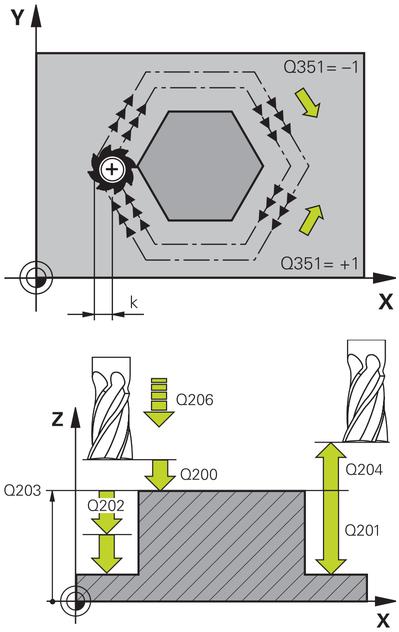
Input: **-99999.9999...+99999.9999**

Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

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

Help graphic



Parameter

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

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

+1 = climb milling

-1 = up-cut milling

PREDEF: The control uses the value of a **GLOBAL DEF** block

(If you enter 0, climb milling is performed)

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

Q201 Depth?

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

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

Q202 Plunging depth?

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

Input: **0...99999.9999**

Q206 Feed rate for plunging?

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

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

Q200 Set-up clearance?

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

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

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. 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 only executed 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 OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q206=+3000              ;FEED RATE FOR PLNGNG ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q385=+500               ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

6.9 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

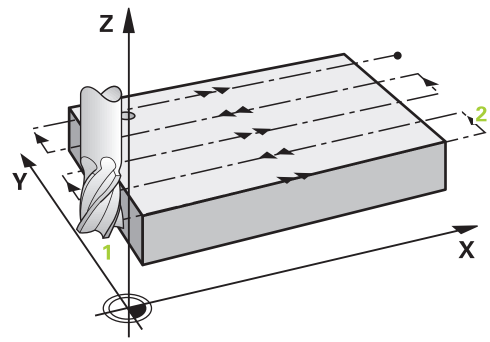
Related topics

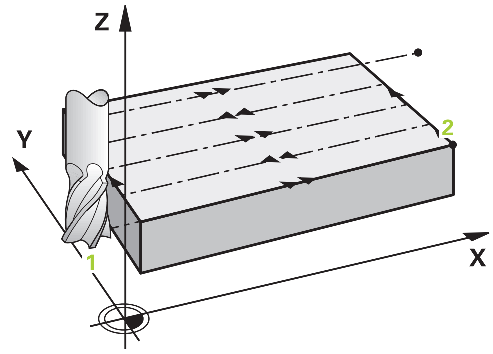
- Cycle **232 FACE MILLING**

  Further information: "Cycle 232 FACE MILLING ", Page 476

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.



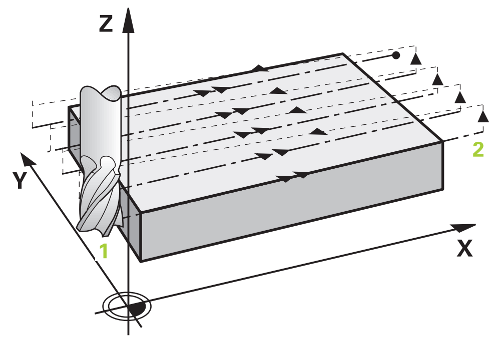


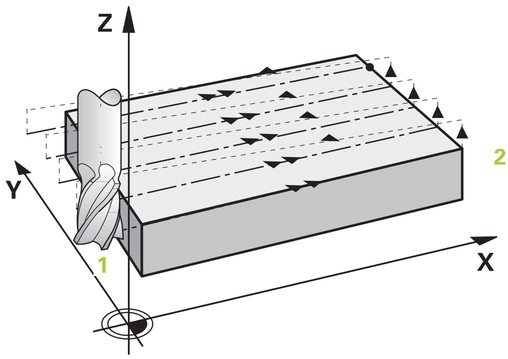
Cycle sequence

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.



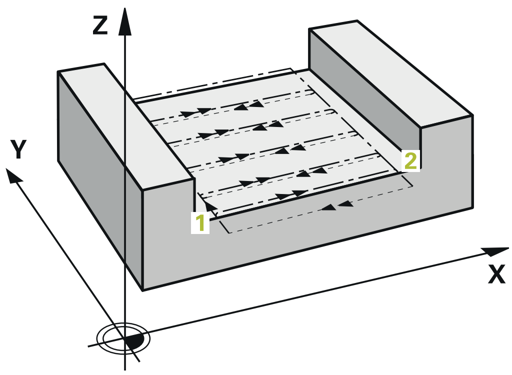


Cycle sequence

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:

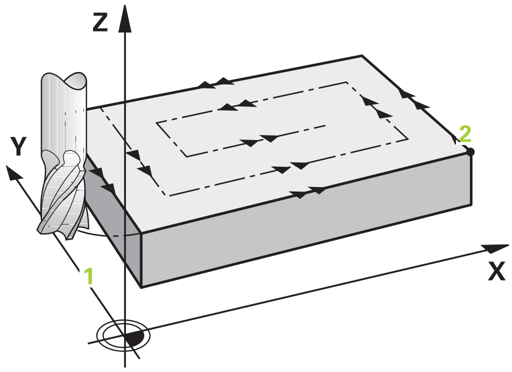


1 The control positions the tool at **FMAX** to the starting point in the working plane. This position is offset from the edge of the workpiece by the tool radius and the set-up clearance **Q357** to the side.
2 The tool moves at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** 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

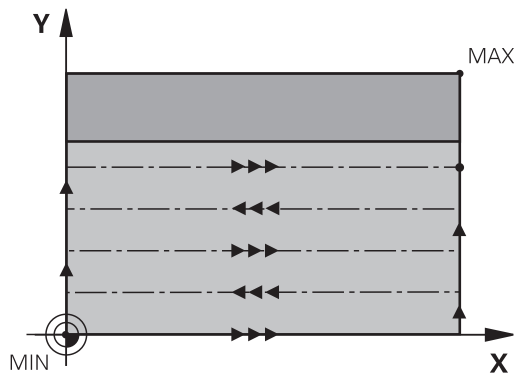
Cycle sequence



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

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 only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- Cycle **233** monitors the entries made for the tool or cutting edge length in **LCUTS** in the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.
- 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**.

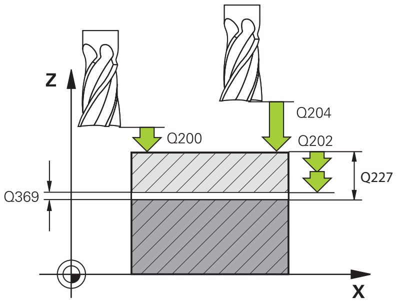
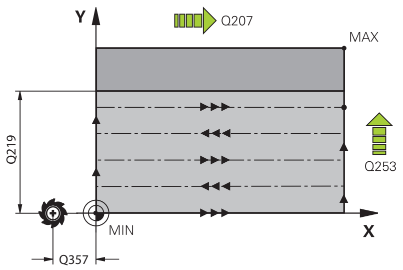
Notes on programming

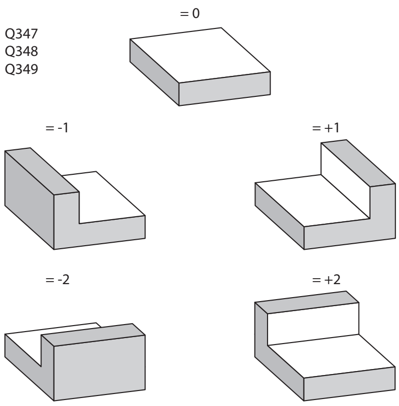
- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- If 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 only executed 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**
	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**

Help graphic	Parameter
	**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**

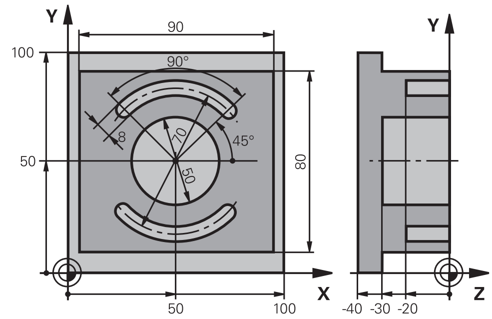
Help graphic	Parameter
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q367 Surface position (-1/0/1/2/3/4)? 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                  ;MACHINING OPERATION ~
    Q389=+2                  ;MILLING STRATEGY ~
    Q350=+1                  ;MILLING DIRECTION ~
    Q218=+60                 ;FIRST SIDE LENGTH ~
    Q219=+20                 ;2ND SIDE LENGTH ~
    Q227=+0                  ;STARTNG PNT 3RD AXIS ~
    Q386=+0                  ;END POINT 3RD AXIS ~
    Q369=+0                  ;ALLOWANCE FOR FLOOR ~
    Q202=+5                  ;MAX. PLUNGING DEPTH ~
    Q370=+1                  ;TOOL PATH OVERLAP ~
    Q207=+500                ;FEED RATE MILLING ~
    Q385=+500                ;FINISHING FEED RATE ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q357=+2                  ;CLEARANCE TO SIDE ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q347=+0                  ;1ST LIMIT ~
    Q348=+0                  ;2ND LIMIT ~
    Q349=+0                  ;3RD LIMIT ~
    Q220=+0                  ;CORNER RADIUS ~
    Q368=+0                  ;ALLOWANCE FOR SIDE ~
    Q338=+0                  ;INFEED FOR FINISHING ~
    Q367=-1                  ;SURFACE POSITION
12 L  X+50  Y+50 R0 FMAX M99
```

6.10 Programming examples

Example: Milling pockets, studs and slots



0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 6 Z S3500	; Tool call: roughing/finishing
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+90 ;FIRST SIDE LENGTH ~	
Q424=+100 ;WORKPC. BLANK SIDE 1 ~	
Q219=+80 ;2ND SIDE LENGTH ~	
Q425=+100 ;WORKPC. BLANK SIDE 2 ~	
Q220=+0 ;CORNER RADIUS ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q437=+0 ;APPROACH POSITION ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q338=+10 ;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 ;MACHINING OPERATION ~	
Q223=+50 ;CIRCLE DIAMETER ~	

Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q366=+1 ;PLUNGE ~	
Q385=+750 ;FINISHING FEED RATE ~	
Q439=+0 ;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99	; Cycle call for circular pocket
9 TOOL CALL 3 Z S5000	; Tool call: slot milling cutter
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 254 CIRCULAR SLOT ~	
Q215=+0 ;MACHINING OPERATION ~	
Q219=+8 ;SLOT WIDTH ~	
Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q375=+70 ;PITCH CIRCLE DIAMETR ~	
Q367=+0 ;REF. SLOT POSITION ~	
Q216=+50 ;CENTER IN 1ST AXIS ~	
Q217=+50 ;CENTER IN 2ND AXIS ~	
Q376=+45 ;STARTING ANGLE ~	
Q248=+90 ;ANGULAR LENGTH ~	
Q378=+180 ;STEPPING ANGLE ~	
Q377=+2 ;NR OF REPETITIONS ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-20 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q366=+2 ;PLUNGE ~	
Q385=+500 ;FINISHING FEED RATE ~	
Q439=+0 ;FEED RATE REFERENCE	

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

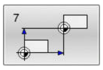
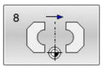
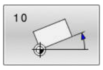
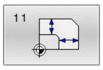
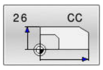
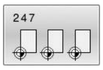
7

Cycles: Coordinate Transformations

7.1 Fundamentals

Overview

Once a contour has been programmed, the control can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The control provides the following functions for coordinate transformations:

Soft key	Cycle	Page
	Cycle 7 DATUM SHIFT ■ Shifting contours directly in the NC program ■ Or shifting contours using datum tables	241
	Cycle 8 MIRRORING ■ Mirroring contours	244
	Cycle 10 ROTATION ■ Rotating contours in the working plane	245
	Cycle 11 SCALING FACTOR ■ Resizing contours	247
	Cycle 26 AXIS-SPECIFIC SCALING ■ Axis-specific resizing of contours	248
	Cycle 19 WORKING PLANE (option 8) ■ Executing machining operations in a tilted coordinate system ■ On machines with swivel heads and/or rotary tables	249
	Cycle 247 PRESETTING ■ Datum setting during program run	255

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:

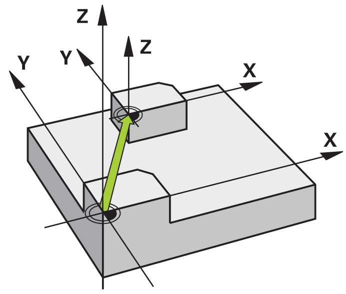
- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

7.2 Cycle 7 DATUM SHIFT

ISO programming
G53/G54

Application

Refer to your machine manual.

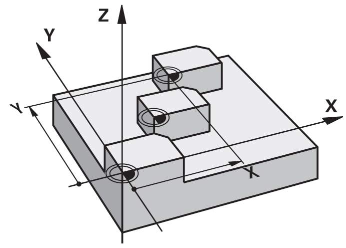


A datum shift allows machining operations to be repeated at various locations on the workpiece. Within an NC program, you can either program datum points directly in the cycle definition or call them from a datum table.

Use datum tables for the following purposes:

- Frequent use of the same datum shift
- Frequently recurring machining sequences on different workpieces
- Frequently recurring machining sequences at various locations on one workpiece

After the definition of a datum shift cycle, all coordinate data will reference the new datum. The control displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.



Reset

- To shift the datum back to the coordinates X=0, Y=0 etc., program another cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.

Status display

The additional status display **TRANS** contains the following information:

- Coordinates from the datum shift
- Name and path of the active datum table
- Active datum number for datum tables
- Comment from the **DOC** column of the active datum number from the datum table

Related topics

- Datum shift with **TRANS DATUM**

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

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining mode.
- The main axis, secondary axis and tool axis are in effect in the W-CS or WPL-CS coordinate system. Rotary axes and parallel axes are in effect in the M-CS system.

Notes about machine parameters

- In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

Additional information regarding datum shifts with datum tables:

- Datums from a datum table **always and exclusively** reference the current preset.
- If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.
- If you work without **SEL TABLE,** then you must activate the desired datum table before the test run or the program run (this applies also to the program run):
  - Use the file manager to select the desired table for a test run in the **Test Run** operating mode: The table now has the status S
  - Use the file manager in the **Program run, single block** and **Program run, full sequence** operating modes to select the desired table for program run: The table receives the status M
- The coordinate values from datum tables are only effective with absolute coordinate values.

Cycle parameters

Datum shift without a datum table

Help graphic	Parameter
	Shift? Enter the coordinates of the new datum. Absolute values are referenced to the workpiece datum, which is determined by the presetting. Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted). Up to six NC axes are possible. Input: **-999999999...+999999999**

Example

```
11 CYCL DEF 7.0 DATUM SHIFT
12 CYCL DEF 7.1  X+60
13 CYCL DEF 7.2  Y+40
14 CYCL DEF 7.3  Z+5
```

Datum shift with a datum table

Help graphic	Parameter
	Shift? Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input: **0...9999**

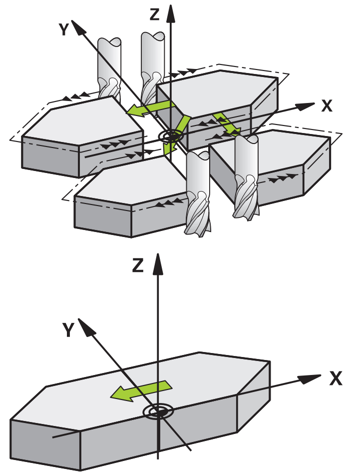
Example

```
11 CYCL DEF 7.0 DATUM SHIFT
12 CYCL DEF 7.1 #5
```

7.3 Cycle 8 MIRRORING

ISO programming
G28

Application



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

Mirroring takes effect as soon as it has been defined in the NC program. It is also in effect in the **Positioning w/ Manual Data Input** operating mode. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed; this does not apply to SL cycles
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.

Reset

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

Related topics

- Mirroring with **TRANS MIRROR**

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

Notes

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

> For working in a tilted system with Cycle **8**, the following procedure is recommended:
> - **First** program the tilting movement and **then** call Cycle **8 MIRRORING**!

Cycle parameters

Help graphic	Parameter
	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. Input: **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

```
11 CYCL DEF 8.0 MIRRORING
12 CYCL DEF 8.1 X Y Z
```

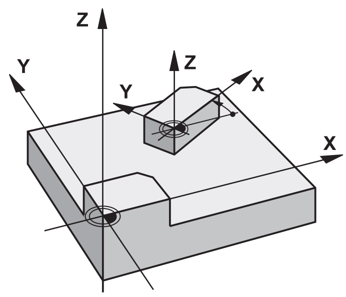
7.4 Cycle 10 ROTATION

ISO programming
G73

Application

Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

The ROTATION cycle takes effect as soon as it has been defined in the NC program. It is also in effect in the **Positioning w/ Manual Data Input** operating mode. The active angle of rotation is shown in the additional status display.



Reference axis for the rotation angle:

- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis

Reset

Program Cycle **10 ROTATION** again and specify a rotation angle of 0°.

Related topics

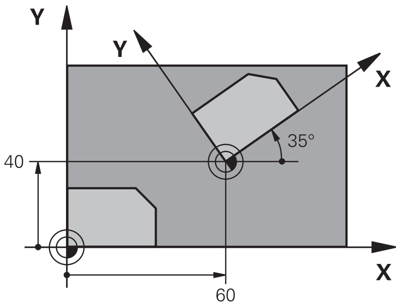
- Rotation with **TRANS ROTATION**

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

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **10** cancels an active radius compensation. If necessary, reprogram the radius compensation.
- After defining Cycle **10**, move both axes of the working plane to activate the rotation for all axes.

Cycle parameters

Help graphic	Parameter
	**Rotation angle?** Enter the angle of rotation in degrees (°). Enter the value as an incremental or absolute value. Input: **-360.000...+360.000**

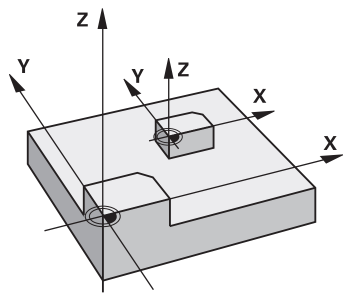
Example

```
11 CYCL DEF 10.0 ROTATION
12 CYCL DEF 10.1  ROT+35
```

7.5 Cycle 11 SCALING FACTOR

ISO programming
G72

Application



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

The scaling factor takes effect as soon as it has been defined in the NC program. It is also in effect in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- all three coordinate axes at the same time
- dimensions in cycles

Requirement

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

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

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

Reset

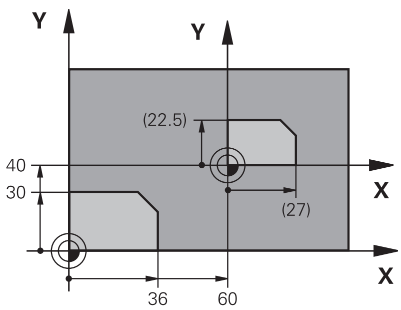
Program Cycle **11 SCALING FACTOR** again and specify a scaling factor of 1.

Related topics

- Scaling with **TRANS SCALE**

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

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

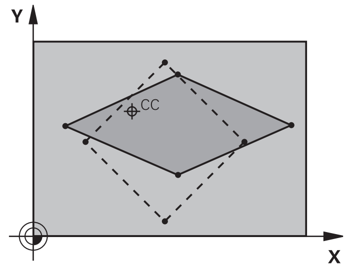
```
11 CYCL DEF 11.0 SCALING FACTOR
12 CYCL DEF 11.1 SCL 0.75
```

7.6 Cycle 26 AXIS-SPECIFIC SCALING

ISO programming

NC syntax is available only in Klartext programming.

Application



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

The scaling factor takes effect as soon as it has been defined in the NC program. It is also in effect in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

Reset

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

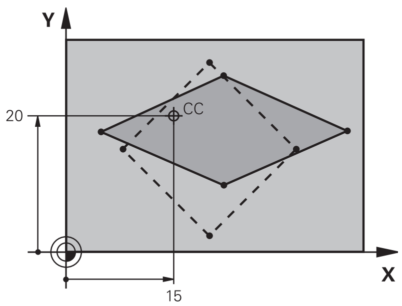
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The contour is enlarged or reduced relative to the center, and not necessarily (as in Cycle **11 SCALING FACTOR**) relative to the active datum.

Notes on programming

- Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.
- You can program each coordinate axis with its own axis-specific scaling factor.
- In addition, you can enter the coordinates of a center for all scaling factors.

Cycle parameters

Help graphic	Parameter
	**Axis and factor?** Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction. Input: **0.000001...99.999999**
	Centerpoint coord. of extension? Center of the axis-specific enlargement or reduction. Input: **-999999999...+999999999**

Example

```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

7.7 Cycle 19 WORKING PLANE (option 8)

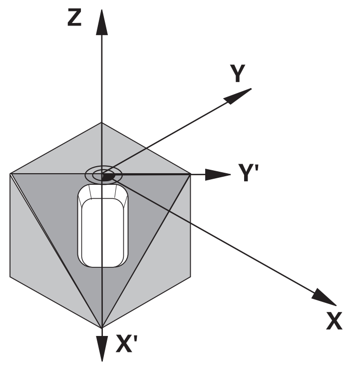
ISO programming
G80

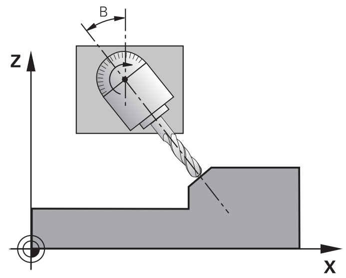
Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Instead of Cycle **19**, HEIDENHAIN recommends programming the more powerful **PLANE** functions.

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





Use Cycle **19** to define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to three rotations (spatial angles) of the **machine-based** coordinate system.

  The required spatial angles can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.

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

If you program the position of the working plane via spatial angles, the control will calculate the required angle positions of the tilted axes automatically and will store these in the **Q120** (A axis) to **Q122** (C axis) parameters. If two solutions are possible, the control will choose the shorter path from the current position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The control first rotates the A axis, then the B axis, and finally the C axis.

Cycle **19** becomes effective as soon as it has been defined in the NC program. As soon as you move an axis in the tilted system, the compensation for this specific axis will be activated. You must move all axes to activate compensation for all axes.

If you have set the **Tilting program run** function to **Active** in the Manual Operation operating mode, the angular value entered in this menu is overwritten by Cycle **19 WORKING PLANE**.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- In combination with a radial facing slide kinematics model, this cycle can also be used in the **FUNCTION MODE TURN** machining mode.
- The working plane is always tilted around the active datum.
- If you use the Cycle **19** while **M120** is active, the control automatically cancels the radius compensation, which also cancels the **M120** function.

Notes on programming

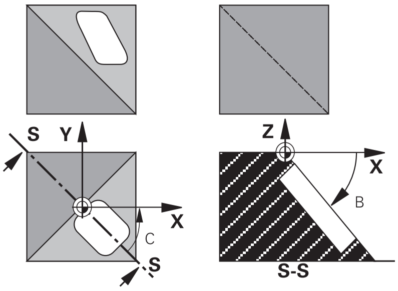
- Write the program as if the machining process was to be executed in a non-tilted plane.
- If you call the cycle again for other angles, you do not need to reset the machining parameters.

Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

Notes about machine parameters

- The machine manufacturer specifies whether the programmed angles are interpreted by the control as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).
- In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

Cycle parameters

Help graphic	Parameter
	**Rotary axis and angle?** Enter the axis of rotation together with the associated tilt angles. Program the rotary axes A, B and C using soft keys. Input: **-360.000...+360.000**

If the control automatically positions the rotary axes, you can enter the following parameters:

Help graphic	Parameter
	Feed rate? F= Traverse speed of the rotary axis during automatic positioning Input: **0...300000**
	Set-up clearance? The control positions the tilting head in such a way that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. This value has an incremental effect. Input: **0...999999999**

Reset

To reset the tilt angles, redefine Cycle **19 WORKING PLANE**. Enter an angular value of 0° for all rotary axes. Then, redefine Cycle **19 WORKING PLANE**. Confirm the dialog prompt by pressing the **NO ENT** key. This disables the function.

Positioning the axes of rotation

Refer to your machine manual.
The machine manufacturer determines whether Cycle **19** positions the axes of rotation automatically or whether they need to be positioned manually in the NC program.

Manual positioning of rotary axes

If Cycle **19** does not position the rotary axes automatically, you need to position them in a separate L block following the cycle definition.

If you use axis angles, you can define the axis values right in the L block. For using spatial angles, program the Q parameters **Q120** (A axis value), **Q121** (B axis value) and **Q122** (C axis value) according to Cycle **19**.

> For manual positioning, always use the rotary axis positions stored in Q parameters **Q120** to **Q122**.
> Avoid the use of functions such as **M94** (modulo rotary axes) in order to prevent discrepancies between actual and nominal positions of the rotary axes for multiple calls.

Example

11 L Z+100 R0 FMAX	
12 L X+25 Y+10 R0 FMAX	
*** - ...**	; Define the spatial angles for calculating the compensation
13 CYCL DEF 19.0 WORKING PLANE	
14 CYCL DEF 19.1 A+0 B+45 C+0	
15 L A+Q120 C+Q122 R0 F1000	; Position the rotary axes by using values calculated by Cycle 19
16 L Z+80 R0 FMAX	; Activate compensation for the spindle axis
17 L X-8.5 Y-10 R0 FMAX	; Activate compensation for the working plane

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle **19**:

- The control can position only closed-loop axes.
- To position the tilted axes, you must enter a feed rate and a set-up clearance, in addition to the tilting angles, when defining the cycle
- Use only preset tools (the full tool length must have been defined)
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting.
- The control performs tilting at the last programmed feed rate (the maximum feed rate depends on the complexity of the swivel head geometry or tilting table)

Example

11 L Z+100 R0 FMAX	
12 L X+25 Y+10 R0 FMAX	
*** - ...**	; Angle for calculating the compensation; define the feed rate and clearance
13 CYCL DEF 19.0 WORKING PLANE	
14 CYCL DEF 19.1 A+0 B+45 C+0 F5000 ABST50	
15 L Z+80 R0 FMAX	; Activate compensation for the spindle axis
16 L X-8.5 Y-10 R0 FMAX	; Activate compensation for the working plane

Position display in a tilted system

On activation of Cycle **19**, the displayed positions (**NOML** and **ACTL**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. This means that the position displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle **19**.

Monitoring of the working space

The control monitors only those axes in the tilted coordinate system that are moved. Where applicable, the control displays an error message.

Positioning in a tilted coordinate system

With miscellaneous function **M130**, you can move the tool, while the coordinate system tilted, to positions that reference the non-tilted coordinate system.

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

- Positioning is without length compensation.
- Positioning is done without length compensation.
- Tool radius compensation is not allowed.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is tilted about the active datum. You can program a datum shift before activating Cycle **19**. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after the activation of Cycle **19** , you are shifting the tilted coordinate system.

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

1. Activate datum shift
2. Activate **Tilt working plane**
3. Activate rotation

...

Workpiece machining

...

1. Reset the rotation
2. Reset **Tilt working plane**
3. Reset the datum shift

Procedure for working with Cycle 19 WORKING PLANE

Proceed as follows:

- Create the NC program
- Clamp the workpiece
- Set any presets
- Start the NC program

Creating the NC program:

- Call the defined tool
- Retract in the spindle axis
- Position the axes of rotation
- Activate a datum shift if required
- Define Cycle **19 WORKING PLANE**
- Position all principal axes (X, Y, Z) in order to activate the compensation
- Define Cycle **19** with different angles, if necessary
- Reset Cycle **19** by programming 0° for all rotary axes
- Redefine Cycle **19** in order to deactivate the working plane
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

You can define the preset in the following ways:

- Manually by touch-off
- Controlled with a HEIDENHAIN 3D touch probe
- Automatically with a HEIDENHAIN 3D touch probe

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

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

7.8 Cycle 247 PRESETTING

ISO programming
G247

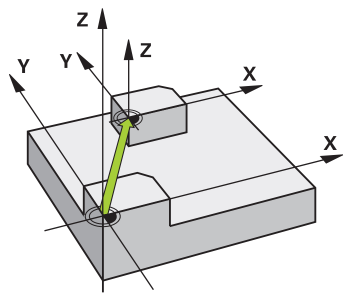
Application

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

After cycle definition, all coordinate input and datum shifts (absolute or incremental) reference the new preset.

Status display

In the status display; the control shows the active preset number behind the preset symbol.



Related topics

- Activate the preset
  Further information: User's Manual for **Klartext Programming**
- Copy the preset
  Further information: User's Manual for **Klartext Programming**
- Correct the preset
  Further information: User's Manual for **Klartext Programming**
- Setting and activating presets
  Further information: User's Manual for **Setup, Testing and Running NC Programs**

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**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** 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** or **Electronic handwheel** operating mode.
- Cycle **247** is also in effect in the Test Run operating mode.

Cycle parameters

Help graphic	Parameter
	Number for preset? Enter the number of the desired preset from the preset table. Alternatively, you can use the **SELECT** soft key to directly select the desired preset from the preset table. Input: **0...65535**

Example

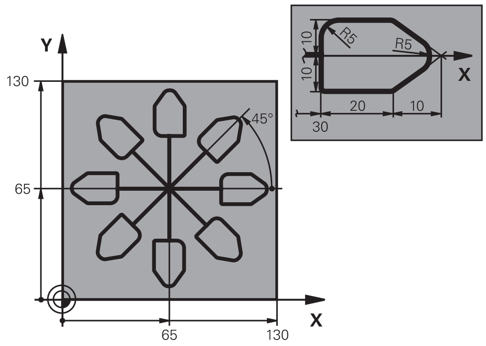
```
11 CYCL DEF 247 PRESETTING ~
    Q339=+4                    ;PRESET NUMBER
```

7.9 Programming examples

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
17 LBL 1	; Subprogram 1
18 L X+0 Y+0 R0 FMAX	; Define milling operation
19 L Z+2 R0 FMAX	
20 L Z-5 R0 F200	
21 L X+30 RL	
22 L IY+10	
23 RND R5	
24 L IX+20	
25 L IX+10 IY-10	
26 RND R5	
27 L IX-10 IY-10	
28 L IX-10 IY-10	
29 L IX-20	
30 L IY+10	

31 L X+0 Y+0 R0 F5000	
32 L Z+20 R0 FMAX	
33 LBL 0	
34 END PGM C220 MM	

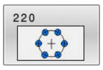
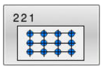
8

Cycles: Pattern Definitions

8.1 Fundamentals

Overview

The control provides three cycles for machining point patterns:

Soft key	Cycle	Page
	Cycle 220 POLAR PATTERN ■ Defining a circular pattern ■ Full circle or pitch circle ■ Input of start and end angles	262
	Cycle 221 CARTESIAN PATTERN ■ Defining a linear pattern ■ Input of an angle of rotation	266
	Cycle 224 DATAMATRIX CODE PATTERN ■ Converting text to a DataMatrix code to be used as a point pattern ■ Input of position and size	270

You can combine the following cycles with point pattern cycles:

	Cycle 220	Cycle 221	Cycle 224
200 DRILLING	✓	✓	✓
201 REAMING	✓	✓	✓
202 BORING	✓	✓	–
203 UNIVERSAL DRILLING	✓	✓	✓
204 BACK BORING	✓	✓	–
205 UNIVERSAL PECKING	✓	✓	✓
206 TAPPING	✓	✓	–
207 RIGID TAPPING	✓	✓	–
208 BORE MILLING	✓	✓	✓
209 TAPPING W/ CHIP BRKG	✓	✓	–
240 CENTERING	✓	✓	✓
251 RECTANGULAR POCKET	✓	✓	✓
252 CIRCULAR POCKET	✓	✓	✓
253 SLOT MILLING	✓	✓	–
254 CIRCULAR SLOT	–	✓	–
256 RECTANGULAR STUD	✓	✓	–
257 CIRCULAR STUD	✓	✓	–
262 THREAD MILLING	✓	✓	–
263 THREAD MLLNG/CNTSNKG	✓	✓	–
264 THREAD DRILLNG/MLLNG	✓	✓	–
265 HEL. THREAD DRLG/MLG	✓	✓	–
267 OUTSIDE THREAD MLLNG	✓	✓	–

> If you have to machine irregular point patterns, use **CYCL CALL PAT** to develop point tables.
> More regular point patterns are available with the **PATTERN DEF** function.

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

Further information: "Pattern definition with PATTERN DEF", Page 62

8.2 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.

Related topics

- Defining a full circle with **PATTERN DEF**
  Further information: "Defining a full circle", Page 70
- Defining a circle segment with **PATTERN DEF**
  Further information: "Defining a pitch circle", Page 71

Cycle run

1. The control moves the tool at rapid traverse from its current position to the starting point for the first machining operation.
   Sequence:
   - Move to 2nd set-up clearance (spindle axis)
   - Approach the starting point in the working plane
   - Move to set-up clearance above the workpiece surface (spindle axis)
2. From this position, the control executes the last defined fixed machining cycle
3. The tool then approaches the starting point for the next machining operation on a straight line or a circular arc. The tool stops at the set-up clearance (or the 2nd set-up clearance)
4. This procedure (steps 1 to 3) will be repeated until all machining operations have been completed

> If you run this cycle in 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).

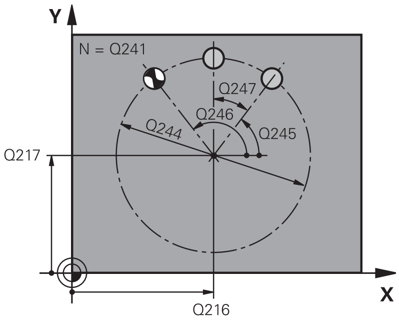
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

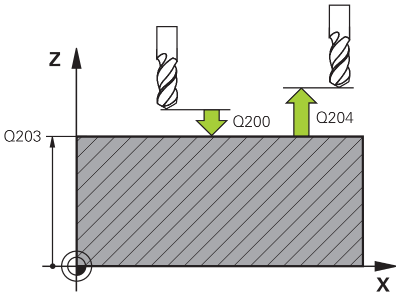
Note on programming

- 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)? Specify how the tool moves between machining processes: **0**: Move to the set-up clearance between operations **1**: Move to the 2nd set-up clearance between operations Input: **0**, **1**
	Q365 Type of traverse? Line=0/arc=1 Specify how the tool moves between machining processes: **0**: Move between operations on a straight line **1**: Move between operations on the pitch circle Input: **0**, **1**

Example

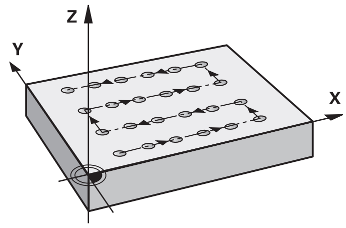
```
11 CYCL DEF 220 POLAR PATTERN ~
    Q216=+50              ;CENTER IN 1ST AXIS ~
    Q217=+50              ;CENTER IN 2ND AXIS ~
    Q244=+60              ;PITCH CIRCLE DIAMETR ~
    Q245=+0               ;STARTING ANGLE ~
    Q246=+360             ;STOPPING ANGLE ~
    Q247=+0               ;STEPPING ANGLE ~
    Q241=+8               ;NR OF REPETITIONS ~
    Q200=+2               ;SET-UP CLEARANCE ~
    Q203=+0               ;SURFACE COORDINATE ~
    Q204=+50              ;2ND SET-UP CLEARANCE ~
    Q301=+1               ;MOVE TO CLEARANCE ~
    Q365=+0               ;TYPE OF TRAVERSE
12 CYCL CALL
```

8.3 Cycle 221 CARTESIAN PATTERN

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.



Related topics

- Defining an individual row with **PATTERN DEF**
  Further information: "Defining a single row", Page 65
- Defining an individual pattern with **PATTERN DEF**
  Further information: "Defining an individual pattern", Page 66

Cycle run

1 The control automatically moves the tool from its current position to the starting point for the first machining operation
  Sequence:
  - Move to 2nd set-up clearance (spindle axis)
  - Approach the starting point in the working plane
  - Move to set-up clearance above the workpiece surface (spindle axis)
2 From this position, the control executes the last defined fixed machining cycle
3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance)
4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first line have been completed. The tool is located above the last point of the first line
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation
6 From this position, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis.
7 This procedure (step 6) will be repeated until all machining operations of the second line have been completed
8 The tool then moves to the starting point of the next row
9 All subsequent lines are machined in a reciprocating movement.

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

Notes

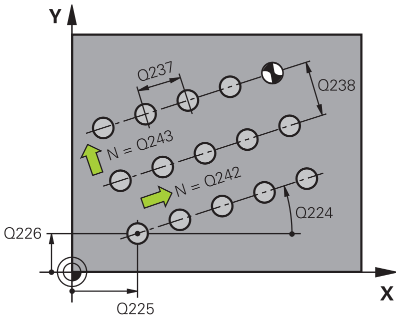
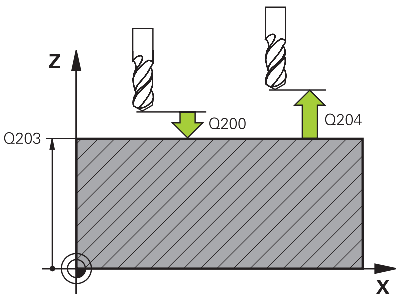
Cycle **221 CARTESIAN PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined 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**
	**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
	Q301 Move to clearance height (0/1)? Specify how the tool moves between machining processes: **0**: Move to the set-up clearance between operations **1**: Move to the 2nd set-up clearance between operations Input: **0**, **1**

Example

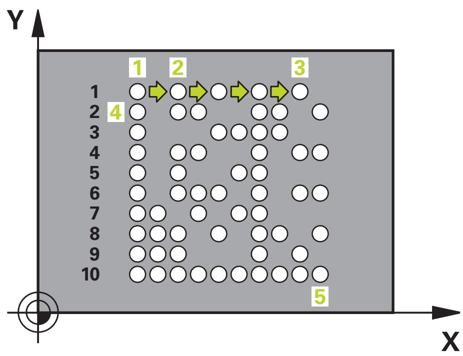
```
11 CYCL DEF 221 CARTESIAN PATTERN ~
    Q225=+15            ;STARTNG PNT 1ST AXIS ~
    Q226=+15            ;STARTNG PNT 2ND AXIS ~
    Q237=+10            ;SPACING IN 1ST AXIS ~
    Q238=+8             ;SPACING IN 2ND AXIS ~
    Q242=+6             ;NUMBER OF COLUMNS ~
    Q243=+4             ;NUMBER OF LINES ~
    Q224=+15            ;ANGLE OF ROTATION ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q301=+1             ;MOVE TO CLEARANCE
12 CYCL CALL
```

8.4 Cycle 224 DATAMATRIX CODE PATTERN

ISO programming
G224

Application

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



Cycle sequence

1 The control automatically moves the tool from its current position to the programmed starting point. This point is always located in the lower left corner.
  Sequence:
  - Move to 2nd set-up clearance (spindle axis)
  - Approach the starting point in the working plane
  - Move to **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 point **1** in the first row
3 From this position, the control executes the last defined fixed machining cycle
4 Then, the control moves the tool in the positive direction of the principal axis to point **2** for the next operation.
5 This procedure will be repeated until all machining operations in the first row have been completed. The tool is located above the last point **3** of the first row
6 Then, the control moves the tool in the negative direction of the principal and secondary axes to the first point **4** of the next row
7 Then, the next points are machined
8 These steps are repeated until the entire DataMatrix code has been completed. Machining stops in the lower right corner **5**
9 Finally, the control retracts the tool to the programmed 2nd set-up clearance

Notes

NOTICE

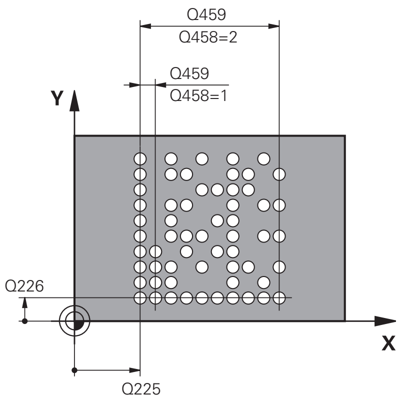
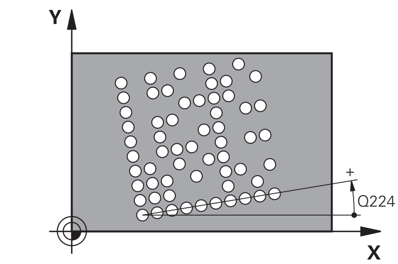
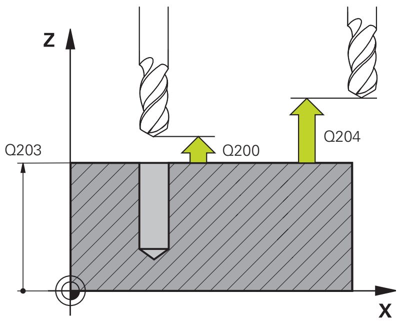
Danger of collision!

If you combine Cycle **224** with one of the machining cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle **224** will be effective for the selected machining cycle. There is a danger of collision!

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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **224** is DEF-active. In addition, Cycle **224** automatically calls the last defined machining cycle.
- The control uses the special character **%** for special functions. If you want to use this character in a DataMatrix code, enter it twice in the text (e.g., **%%**).

Cycle parameters

Help graphic	Parameter
	**Q225 Starting point in 1st axis?** Coordinate in the lower left corner of the code in the main axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate in the bottom left corner of the data matrix code in the secondary axis. 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 273 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**

Example

```
11 CYCL DEF 224 DATAMATRIX CODE PATTERN ~
    Q225=+0              ;STARTNG PNT 1ST AXIS ~
    Q226=+0              ;STARTNG PNT 2ND AXIS ~
    QS501=""             ;TEXT ~
    Q458=+1              ;SIZE SELECTION ~
    Q459=+1              ;SIZE ~
    Q224=+0              ;ANGLE OF ROTATION ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

Outputting variable texts in DataMatrix codes

In addition to specified characters you can also output certain variables in DataMatrix codes. Precede the variable with **%**.

You can use the following variable texts in Cycle **224 DATAMATRIX CODE PATTERN**:

- Date and time
- Names and paths of NC programs
- Count values

Date and time

You can convert the current date, the current time, or the current calendar week into a DataMatrix code. Enter the value **%time<x>** in cycle parameter **QS501**. **<x>** defines the format (e.g., 08 for DD.MM.YYYY.)

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9 (e.g., **%time08**).

The following formats are available:

Input	Format
%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 in the MOD menu.

Enter the value **%count<x>** in cycle parameter **QS501**.

The number after **%count** indicates how many digits the DataMatrix code contains. The maximum is nine digits.

Example:

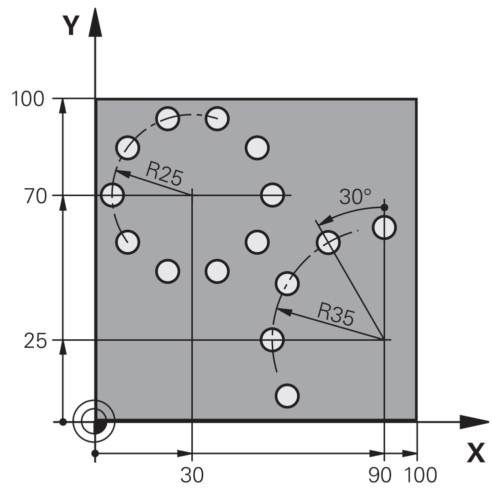
- Programming: **%count9**
- Current count value: 3
- Result: 000000003

Operating information

- In the Test Run operating mode, the control only simulates the counter reading that you define directly in the NC program. The counter reading from the MOD menu is ignored.
- In the SINGLE BLOCK and FULL SEQ. operating modes, the control will take the counter reading from the MOD menu into account.

8.5 Programming examples

Example: Polar hole patterns



0 BEGIN PGM 200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 200 Z S3500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-15 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+4 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.25 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
6 CYCL DEF 220 POLAR PATTERN ~	
Q216=+30 ;CENTER IN 1ST AXIS ~	
Q217=+70 ;CENTER IN 2ND AXIS ~	
Q244=+50 ;PITCH CIRCLE DIAMETR ~	
Q245=+0 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+0 ;STEPPING ANGLE ~	
Q241=+10 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;MOVE TO CLEARANCE ~	
Q365=+0 ;TYPE OF TRAVERSE	

7 CYCL DEF 220 POLAR PATTERN ~	
Q216=+90 ;CENTER IN 1ST AXIS ~	
Q217=+25 ;CENTER IN 2ND AXIS ~	
Q244=+70 ;PITCH CIRCLE DIAMETR ~	
Q245=+90 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+30 ;STEPPING ANGLE ~	
Q241=+5 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;MOVE TO CLEARANCE ~	
Q365=+0 ;TYPE OF TRAVERSE	
8 L Z+100 R0 FMAX	; Retract the tool
9 M30	; End of program
10 END PGM 200 MM	

9

Cycles: Contour Pocket

9.1 SL Cycles

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 Optimized Contour Milling function (option 167).

Related topics

- Optimized Contour Milling (option 167)

  Further information: "Cycles: Optimized Contour Milling", Page 329

> Programming and operating notes:
> - The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
> - SL Cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always use a graphic test run to verify your program before running it. This is a simple way of finding out whether the program calculated by the control will provide the desired results.
> - If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Characteristics of the subprograms

- Closed contour without approach and departure movements
- Coordinate transformations are permitted; if they are programmed within the subcontours, they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The control recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR
- The control recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL
- The subprograms must not contain spindle axis coordinates.
- Always program both axes in the first NC block of the subprogram
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms
- Without machining cycles, feed rates, and M functions

Cycle properties

- The control automatically positions the tool to the set-up clearance before each cycle. You must move the tool to a safe position before the cycle call
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling

The machining data, such as milling depth, allowances, and set-up clearance can be entered centrally in Cycle **20 CONTOUR DATA**.

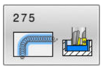
Program structure: Machining with SL Cycles

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

Overview

Soft key	Cycle	Page
	Cycle 14 CONTOUR ■ Listing the contour subprograms	283
	Cycle 20 CONTOUR DATA ■ Input of machining information	287
	Cycle 21 PILOT DRILLING ■ Machining a hole for non-center cutting tools	290
	Cycle 22 ROUGH-OUT ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	292
	Cycle 23 FLOOR FINISHING ■ Finishing with finishing allowance for the floor from Cycle **20**	297
	Cycle 24 SIDE FINISHING ■ Finishing with side finishing allowance from Cycle **20**	300

Enhanced cycles:

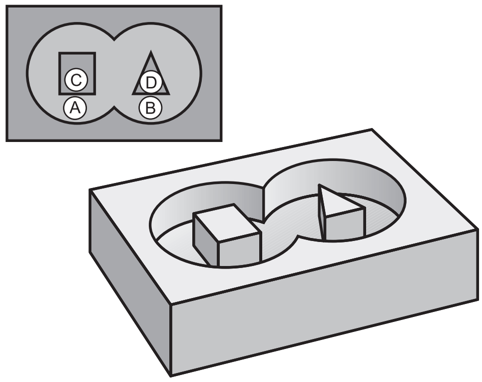
Soft key	Cycle	Page
	Cycle 270 CONTOUR TRAIN DATA ■ Input of contour data for Cycle **25** or **276**	304
	Cycle 25 CONTOUR TRAIN ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	306
	Cycle 275 TROCHOIDAL SLOT ■ Machining of open and closed slots using trochoidal milling.	311
	Cycle 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	317

9.2 Cycle 14 CONTOUR

ISO programming
G37

Application

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



Related topics

- Simple contour formula

  Further information: "SL or OCM cycles with simple contour formula", Page 437
- Complex contour formula

  Further information: "SL or OCM cycles with complex contour formula", Page 426

Notes

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

Cycle parameters

Help graphic	Parameter
	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**

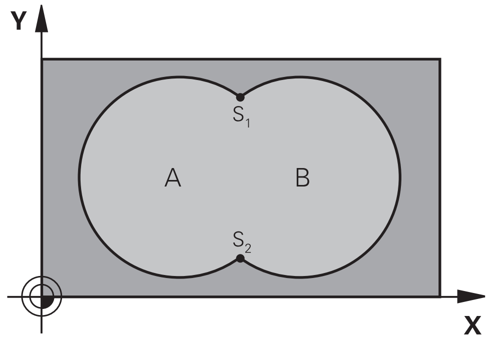
Example

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

9.3 Superimposing contours

Fundamentals

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



Subprograms: overlapping pockets

> The following examples show contour subprograms that are called by Cycle **14 CONTOUR** in a main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2. They need not be programmed.

The pockets are programmed as full circles.

Subprogram 1: Pocket A

11 LBL 1
12 L X+10 Y+10 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0

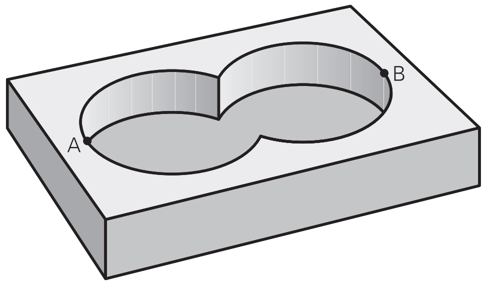
Subprogram 2: Pocket B

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

Surface resulting from sum

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

- The surfaces A and B must be pockets
- The first pocket (in Cycle **14**) must start outside the second pocket



Surface A:

```
11 LBL 1
12 L  X+10  Y+50 RR
13 CC  X+35  Y+50
14 C  X+10  Y+50 DR-
15 LBL 0
```

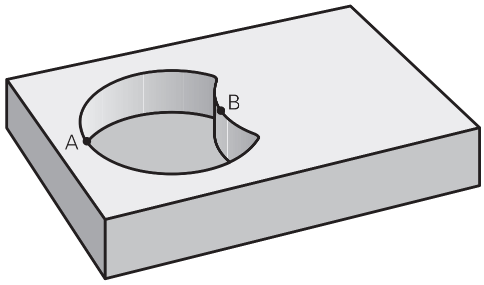
Surface B:

```
16 LBL 2
17 L  X+90  Y+50 RR
18 CC  X+65  Y+50
19 C  X+90  Y+50 DR-
20 LBL 0
```

Surface resulting from difference

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

- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.



Surface A:

```
11 LBL 1
12 L  X+10  Y+50 RR
13 CC  X+35  Y+50
14 C  X+10  Y+50 DR-
15 LBL 0
```

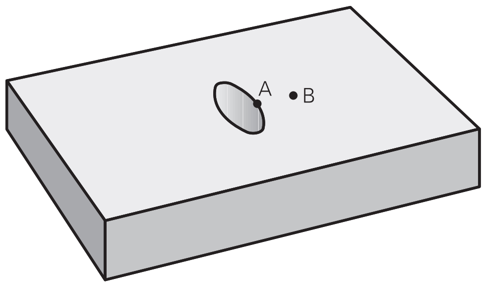
Surface B:

```
16 LBL 2
17 L  X+40  Y+50 RL
18 CC  X+65  Y+50
19 C  X+40  Y+50 DR-
20 LBL 0
```

Surface resulting from intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- A and B must be pockets
- A must start inside of B



Surface A:

```
11 LBL 1
12 L  X+60  Y+50 RR
13 CC  X+35  Y+50
14 C  X+60  Y+50 DR-
15 LBL 0
```

Surface B:

```
16 LBL 2
17 L  X+90  Y+50 RR
18 CC  X+65  Y+50
19 C  X+90  Y+50 DR-
20 LBL 0
```

9.4 Cycle 20 CONTOUR DATA

ISO programming

G120

Application

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

Related topics

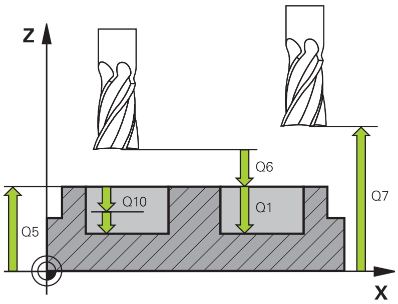
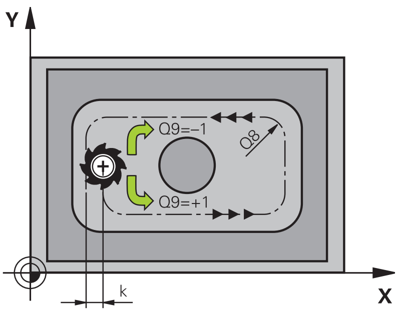
- Cycle **271 OCM CONTOUR DATA** (option 167)

  Further information: "Cycle 271 OCM CONTOUR DATA (option 167) ", Page 338

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **20** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **20** are valid for Cycles **21** to **24**.
- If you are using the SL cycles in **Q** parameter programs, the cycle parameters **Q1** to **Q20** cannot be used as program parameters.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH = 0, the control performs the cycle at the depth 0.

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between workpiece surface and pocket floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q2 Path overlap factor? **Q2** 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**
	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               ;ROTATIONAL DIRECTION
```

9.5 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).

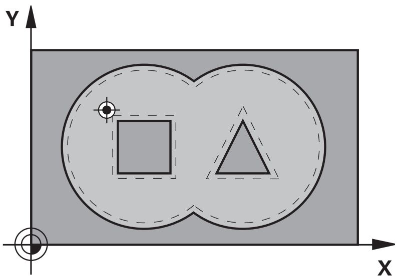
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.
- In narrow areas, the control may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.
- If **Q13**=0, the control uses the data of the tool that is currently in the spindle.

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining. After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have programmed **ToolAxClearanceHeight**.

Cycle parameters

Help graphic	Parameter
	**Q10 Plunging depth?** Tool infeed per cut (minus sign for negative machining direction). This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q13 or **QS13 Rough-out tool number/name?** Number or name of the rough-out tool. You are able to transfer the tool directly from the tool table via soft key. Input: **0...999999.9** or max. **255** characters

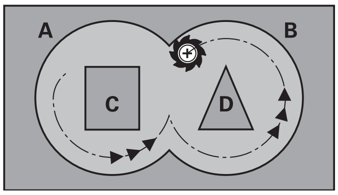
Example

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

9.6 Cycle 22 ROUGH-OUT

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** (option 167)
  Further information: "Cycle 272 OCM ROUGHING (option 167)", Page 341

Cycle run

1 The control positions the tool above the cutter infeed point, taking the finishing allowance for side into account
2 After reaching the first plunging depth, the tool mills the contour in an outward direction at the programmed milling feed rate **Q12**
3 The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B)
4 In the next step, the control moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached
5 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- During fine roughing, the control does not take a defined wear value **DR** of the coarse roughing tool into account.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q1**, the control will display an error message.
- 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: User's Manual for **Klartext Programming**

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

Notes on programming

- If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.
- Define the plunging behavior of Cycle **22** with parameter **Q19** and in the **ANGLE** and **LCUTS** columns of the tool table:
  - If **Q19** = 0 is defined, the tool will always plunge perpendicularly, even if a plunge angle (**ANGLE**) was defined for the active tool
  - If you define **ANGLE** = 90°, the control will plunge perpendicularly. The reciprocation feed rate **Q19** is used as plunging feed rate
  - If the reciprocation feed rate **Q19** is defined in Cycle **22** and **ANGLE** is between 0.1 and 89.999 in the tool table, the tool plunges helically using the defined **ANGLE**
  - If the reciprocation feed is defined in Cycle **22** and no **ANGLE** is defined in the tool table, the control displays an error message
  - If the geometry conditions do not allow helical plunging (slot geometry), the control tries a reciprocating plunge (the reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**))

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	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?** Number or name of the tool with which the control has already coarse-roughed the contour. You can use the soft key to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself using the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters
	Q19 Feed rate for reciprocation? Reciprocation feed rate in mm/min Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? Tool traversing speed in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q12**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q401 Feed rate factor in %? Percentage value to which the control reduces the machining feed rate (**Q12**) as soon as the tool moves with its entire circumference within the material during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (**Q2**) specified in Cycle **20**. The control then reduces the feed rate as per your definition at transitions and narrow places, reducing the total machining time. Input: **0.0001...100**
	Q404 Fine roughing strategy (0/1)? Define how the control 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             ;FINE ROUGH STRATEGY
```

9.7 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** (option 167)
  Further information: "Cycle 273 OCM FINISHING FLOOR (option 167)", Page 356

Cycle run

1 The control positions the tool to the clearance height at rapid traverse FMAX.
2 The tool then moves in the tool axis at the feed rate **Q11**.
3 The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically
4 The tool clears the finishing allowance remaining from rough-out.
5 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This 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

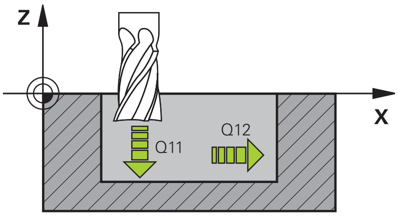
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.
- The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.
- 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: User's Manual for **Klartext Programming**

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	**Q11 Feed rate for plunging?** Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? Tool traversing speed in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q12**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Example

```
11 CYCL DEF 23 FLOOR FINISHING ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q208=+99999         ;RETRACTION FEED RATE
```

9.8 Cycle 24 SIDE FINISHING

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** (option 167)

  Further information: "Cycle 274 OCM FINISHING SIDE (option 167)", Page 360

Cycle run

1. The control positions the tool above the workpiece surface to the starting point for the approach position. This position in the plane results from a tangential arc on which the control moves the tool when approaching the contour
2. The control then moves the tool to the first plunging depth using the feed rate for plunging
3. The contour is approached on a tangential arc and machined up to the end. Each subcontour is finished separately
4. The tool moves on a tangential helical arc when approaching the finishing contour or retracting from it. The starting height of the helix is 1/25 of the set-up clearance **Q6**, but max. the remaining last plunging depth above the final depth
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- If no allowance was defined in Cycle **20**, the control generates the error message "Tool radius too large."
- If you run Cycle **24** without having roughed out with Cycle **22**, then enter "0" for the radius of the rough mill.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle **20**.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.
- You can execute this cycle using a grinding tool.
- 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: User's Manual for **Klartext Programming**

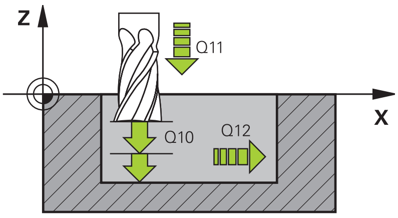
Notes on programming

- The sum of finishing allowance for the side (**Q14**) and the radius of the finish mill must be smaller than the sum of allowance for side (**Q3**, Cycle **20**) and the radius of the rough mill.
- The finishing allowance for the side **Q14** is left over after finishing. Therefore, it must be smaller than the allowance in Cycle **20**.
- Cycle **24** can also be used for contour milling. In that case, you must do the following:
  - Define the contour to be milled as a single island (without pocket boundary)
  - In Cycle **20**, enter a finishing allowance (**Q3**) greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket:
  - **PosBeforeMachining**: Return to starting position.
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	Q9 Direction of rotation? cw = -1 Machining direction: **+1**: Counterclockwise **–1**: Clockwise Input: **-1**, **+1**
	**Q10 Plunging depth?** Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q14 Finishing allowance for side? The finishing allowance for the side **Q14** is left over after finishing. This allowance must be smaller than the allowance in Cycle **20**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. **Q438 = –1**: The control assumes that the tool last used is the rough-out tool (default behavior) **Q438 = 0**: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0. Input: **-1...+32767.9** or **255** characters

Example

```
11 CYCL DEF 24 SIDE FINISHING ~
    Q9=+1              ;ROTATIONAL DIRECTION ~
    Q10=+5             ;PLUNGING DEPTH ~
    Q11=+150           ;FEED RATE FOR PLNGNG ~
    Q12=+500           ;FEED RATE F. ROUGHNG ~
    Q14=+0             ;ALLOWANCE FOR SIDE ~
    Q438=-1            ;ROUGH-OUT TOOL
```

9.9 Cycle 270 CONTOUR TRAIN DATA

ISO programming

G270

Application

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

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **270** is DEF-active, which means that it 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

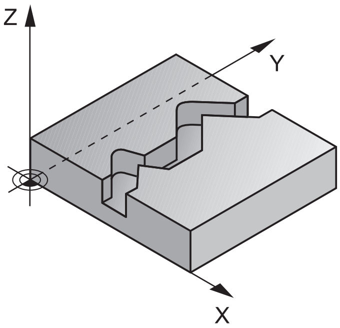
```
11 CYCL DEF 270 CONTOUR TRAIN DATA ~
    Q390=+1                ;TYPE OF APPROACH ~
    Q391=+1                ;RADIUS COMPENSATION ~
    Q392=+5                ;RADIUS ~
    Q393=+90               ;CENTER ANGLE ~
    Q394=+0                ;DISTANCE
```

9.10 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- The control takes only the first label of Cycle **14 CONTOUR** into account.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- You can execute this cycle using a grinding tool.
- 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: User's Manual for **Klartext Programming**

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?** Number or name of the tool with which the control has already coarse-roughed the contour. You can use the soft key to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself using the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters
	Q446 Accepted residual material? Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input: **0...999.999**
	Q448 Path extension? Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour. Input: **0...99.999**

Example

```
11 CYCL DEF 25 CONTOUR TRAIN ~
    Q1=-20                ;MILLING DEPTH ~
    Q3=+0                 ;ALLOWANCE FOR SIDE ~
    Q5=+0                 ;SURFACE COORDINATE ~
    Q7=+50                ;CLEARANCE HEIGHT ~
    Q10=-5                ;PLUNGING DEPTH ~
    Q11=+150              ;FEED RATE FOR PLNGNG ~
    Q12=+500              ;FEED RATE F. ROUGHNG ~
    Q15=+1                ;CLIMB OR UP-CUT ~
    Q18=+0                ;COARSE ROUGHING TOOL ~
    Q446=+0.01            ;RESIDUAL MATERIAL ~
    Q447=+10              ;CONNECTION DISTANCE ~
    Q448=+2               ;PATH EXTENSION
```

9.11 Cycle 275 TROCHOIDAL SLOT

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** (option 45)).

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

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

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

Program structure: Machining with SL Cycles

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

Cycle sequence

Roughing closed slots

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

1 Following the positioning logic, the tool moves to the starting point of the contour description and moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached

Finishing closed slots

5 If a finishing allowance has been defined, the control finishes the slot walls, in multiple infeeds, if so specified. Starting from the defined starting point, the control approaches the slot wall tangentially. Climb or up-cut milling is taken into consideration.

Roughing open slots

The contour description of an open slot must always start with an approach block (**APPR**).

1 Following the positioning logic, the tool moves to the starting point of the machining operation as defined by the parameters in the **APPR** block and plunges vertically to the first plunging depth.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached

Finishing open slots

5 If a finishing allowance has been defined, the control finishes the slot walls (in multiple infeeds if specified). The control approaches the slot wall starting from the defined starting point of the **APPR** block. Climb or up-cut milling is taken into consideration

Notes

NOTICE

Danger of collision!

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

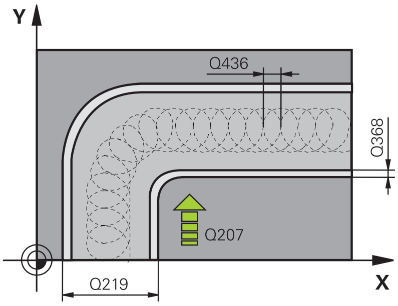
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- In conjunction with Cycle **275**, the control does not require Cycle **20 CONTOUR DATA**.
- This 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.

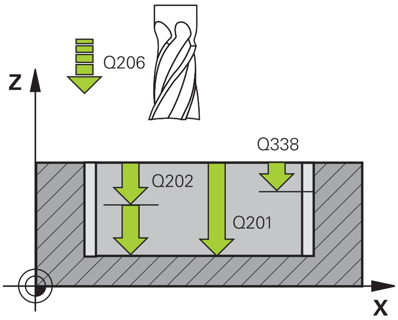
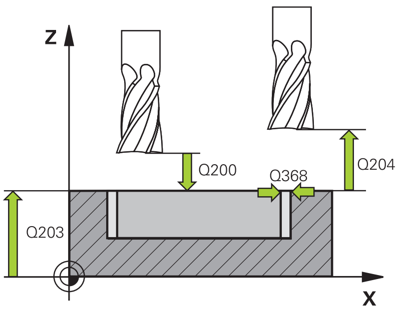
  Further information: User's Manual for **Klartext Programming**

Notes on programming

- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If using Cycle **275 TROCHOIDAL SLOT**, you may define only one contour subprogram in Cycle **14 CONTOUR**.
- Define the center line of the slot with all available path functions in the contour subprogram.
- The starting point of a closed slot must not be located in a contour corner.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q219 Width of slot?** Enter the width of the slot, 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. 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? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

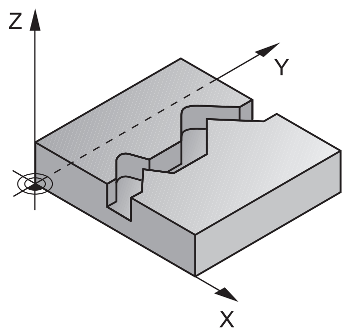
```
11 CYCL DEF 275 TROCHOIDAL SLOT ~
    Q215=+0              ;MACHINING OPERATION ~
    Q219=+10             ;SLOT WIDTH ~
    Q368=+0              ;ALLOWANCE FOR SIDE ~
    Q436=+2              ;INFEED PER REV. ~
    Q207=+500            ;FEED RATE MILLING ~
    Q351=+1              ;CLIMB OR UP-CUT ~
    Q201=-20             ;DEPTH ~
    Q202=+5              ;PLUNGING DEPTH ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q338=+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              ;PLUNGE ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q439=+0              ;FEED RATE REFERENCE
12 CYCL CALL
```

9.12 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 run

Machining a contour without infeed: Milling depth Q1 = 0

1 The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle **270 CONTOUR TRAIN DATA** (e.g., the Type of approach). The control then moves the tool to the first plunging depth
2 According to the previously defined Cycle **270 CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3 At the end of the contour, the tool will be retracted as defined in Cycle **270 CONTOUR TRAIN DATA**
4 Finally, the control retracts the tool to the clearance height.

Machining a contour with infeed: Milling depth Q1 not equal to 0 and plunging depth Q10 are defined

1 The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle **270 CONTOUR TRAIN DATA** (e.g., the Type of approach). The control then moves the tool to the first plunging depth
2 According to the previously defined Cycle **270 CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3 If you selected machining with climb milling and up-cut milling (**Q15** = 0), the control will perform a reciprocation movement. The infeed movement (plunging) will be performed at the end and at the starting point of the contour. If **Q15** is not equal to 0, the tool is moved to clearance height and 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **APPR** and **DEP** blocks for contour approach and departure, the control monitors whether the execution of any of these blocks would damage the contour.
- If using Cycle **25 CONTOUR TRAIN**, you can define only one subprogram in Cycle **14 CONTOUR**.
- We recommend that you use Cycle **270 CONTOUR TRAIN DATA** in conjunction with Cycle **276**. Cycle **20 CONTOUR DATA**, however, is not required.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- 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: User's Manual for **Klartext Programming**

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?** Number or name of the tool with which the control has already coarse-roughed the contour. You can use the soft key to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself using the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters

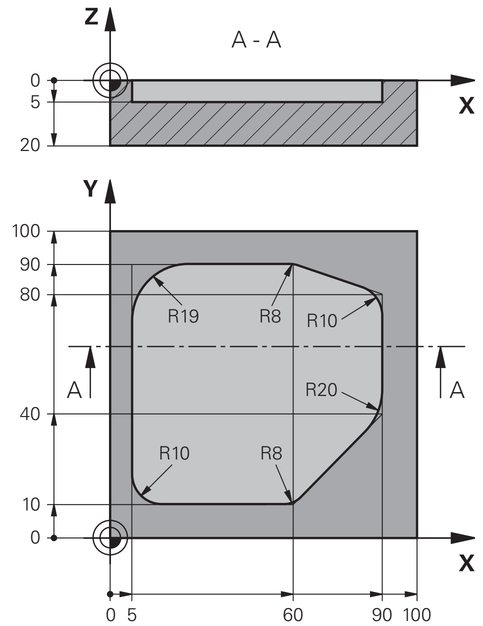
Help graphic	Parameter
	Q446 Accepted residual material? Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input: **0...999.999**
	Q448 Path extension? Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour. Input: **0...99.999**

Example

```
11 CYCL DEF 276 THREE-D CONT. TRAIN ~
    Q1=-20              ;MILLING DEPTH ~
    Q3=+0               ;ALLOWANCE FOR SIDE ~
    Q7=+50              ;CLEARANCE HEIGHT ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q15=+1              ;CLIMB OR UP-CUT ~
    Q18=+0              ;COARSE ROUGHING TOOL ~
    Q446=+0.01          ;RESIDUAL MATERIAL ~
    Q447=+10            ;CONNECTION DISTANCE ~
    Q448=+2             ;PATH EXTENSION
```

9.13 Programming examples

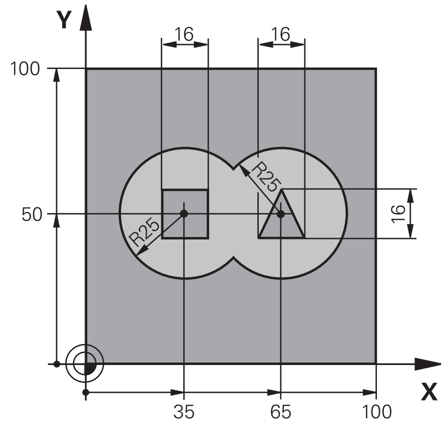
Example: Roughing-out and fine-roughing a pocket with SL Cycles



0 BEGIN PGM 1078634 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 15 Z S4500	; Tool call: coarse roughing tool (diameter: 30)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-5 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q4=+0 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+50 ;CLEARANCE HEIGHT ~	
Q8=+0.2 ;ROUNDING RADIUS ~	
Q9=+1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	

Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+90 ;FEED RATE FACTOR ~	
Q404=+1 ;FINE ROUGH STRATEGY	
9 CYCL CALL	; Cycle call: coarse roughing
10 L Z+200 R0 FMAX	; Retract the tool
11 TOOL CALL 4 Z S3000	; Tool call: fine roughing tool (diameter: 8)
12 L Z+100 R0 FMAX M3	
13 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+15 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	
Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+90 ;FEED RATE FACTOR ~	
Q404=+1 ;FINE ROUGH STRATEGY	
14 CYCL CALL	; Cycle call: fine roughing
15 L Z+200 R0 FMAX	; Retract the tool
16 M30	; End of program
17 LBL 1	; Contour subprogram
18 L X+5 Y+50 RR	
19 L Y+90	
20 RND R19	
21 L X+60	
22 RND R8	
23 L X+90 Y+80	
24 RND R10	
25 L Y+40	
26 RND R20	
27 L X+60 Y+10	
28 RND R8	
29 L X+5	
30 RND R10	
31 L X+5 Y+50	
32 LBL 0	
33 END PGM 1078634 MM	

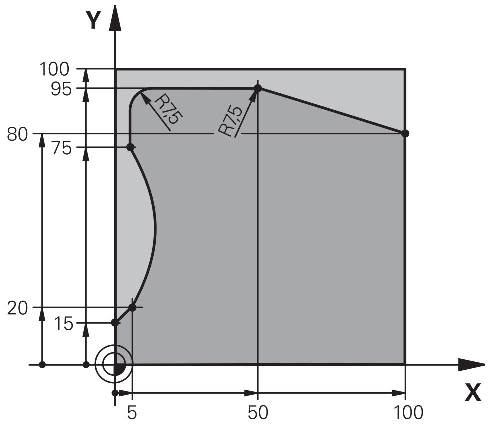
Example: Pilot drilling, roughing and finishing overlapping contours with SL Cycles



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 204 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 ;ROTATIONAL DIRECTION	
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 ;FINE ROUGH STRATEGY	
13 CYCL CALL	; Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+200 ;FEED RATE F. ROUGHNG ~	
Q208=+99999 ;RETRACTION FEED RATE	
15 CYCL CALL	; Cycle call: floor finishing
16 CYCL DEF 24 SIDE FINISHING ~	
Q9=+1 ;ROTATIONAL DIRECTION ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+400 ;FEED RATE F. ROUGHNG ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=-1 ;ROUGH-OUT TOOL	
17 CYCL CALL	; Cycle call: side finishing
18 L Z+100 R0 FMAX	; Retract the tool
19 M30	; End of program
20 LBL 1	; Contour subprogram 1: left pocket
21 CC X+35 Y+50	
22 L X+10 Y+50 RR	
23 C X+10 DR-	
24 LBL 0	
25 LBL 2	; Contour subprogram 2: right pocket
26 CC X+65 Y+50	
27 L X+90 Y+50 RR	
28 C X+90 DR-	
29 LBL 0	
30 LBL 3	; Contour subprogram 3: left square island
31 L X+27 Y+50 RL	
32 L Y+58	
33 L X+43	
34 L Y+42	
35 L X+27	
36 LBL 0	
37 LBL 4	; Contour subprogram 4: right triangular island
38 L X+65 Y+42 RL	
39 L X+57	
40 L X+65 Y+58	
41 L X+73 Y+42	
42 LBL 0	
43 END PGM 2 MM	

Example: Contour train



0 BEGIN PGM 3 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S2000	; Tool call (diameter: 20)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR 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 OR UP-CUT ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q446=+0.01 ;RESIDUAL MATERIAL ~	
Q447=+10 ;CONNECTION DISTANCE ~	
Q448=+2 ;PATH EXTENSION	
8 CYCL CALL	; Cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 M30	; End of program
11 LBL 1	; Contour subprogram
12 L X+0 Y+15 RL	
13 L X+5 Y+20	
13 CT X+5 Y+75	
14 CT X+5 Y+75	
15 L Y+95	
16 RND R7.5	
17 L X+50	

18 RND R7.5	
19 L X+100 Y+80	
20 LBL 0	
21 END PGM 3 MM	

10

Cycles: Optimized Contour Milling

10.1 OCM cycles (option 167)

OCM cycles

General information

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

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

- When roughing, the control will maintain the specified tool angle precisely
- Besides pockets, you can also machine islands and open pockets

Programming and operating notes:

- You can program up to 16 384 contour elements in one OCM cycle.
- OCM cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always perform a graphic test run ! This is a simple way of finding out whether the program calculated by the control will provide the desired results.

Contact angle

When roughing, the control will retain the tool angle precisely. The tool angle can be defined implicitly by specifying an overlap factor. The maximum overlap factor is 1.99; this corresponds to an angle of nearly 180°.

Contour

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

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

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

CONTOUR DEF / SEL CONTOUR:

In **CONTOUR DEF / SEL CONTOUR**, the first contour can be a pocket or a boundary. The next contours can be programmed as islands or pockets. To program open pockets, use a boundary and an island.

Proceed as follows:

- Program **CONTOUR DEF**
- Define the first contour as a pocket and the second one as an island
- Define Cycle **271 OCM CONTOUR DATA**
- Program cycle parameter **Q569 = 1**
- > The control will interpret the first contour as an open boundary instead of a pocket. Thus, the open boundary and the island programmed subsequently are combined to form an open pocket.
- Define Cycle **272 OCM ROUGHING**

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

OCM figure cycles:

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

Proceed as follows:

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

Program structure: Machining with OCM cycles

```
0 BEGIN OCM MM
...
12 CONTOUR DEF
...
13 CYCL DEF 271 OCM CONTOUR DATA
...
16 CYCL DEF 272 OCM ROUGHING
...
17 CYCL CALL
...
20 CYCL DEF 273 OCM FINISHING FLOOR
...
21 CYCL CALL
...
24 CYCL DEF 274 OCM FINISHING SIDE
...
25 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM OCM MM
```

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 394

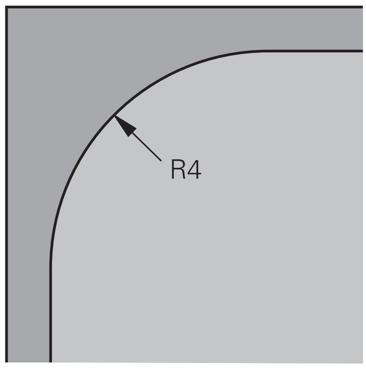
- If residual material remains in the inside corners after roughing, then use a smaller rough-out tool or define an additional roughing operation with a smaller tool.
- If the inside corners cannot be roughed out completely, the control may damage the contour during chamfering. In order to prevent damage 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

Roughing

- ▶ Rough the contour with the tool **MILL_D20_ROUGH**
- > The control takes into account the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 12 mm during initial roughing.

...	
12 TOOL CALL Z "MILL_D20_ROUGH"	
...	
15 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR** **...**	Resulting inside radius = $\mathbf{R_T}$+ (**Q578** * $\mathbf{R_T}$) **10 + (0.2 *10) = 12**
16 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.

...	
20 TOOL CALL Z "MILL_D10_ROUGH"	
...	
22 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR** **...**	Resulting inside radius = $\mathbf{R_T}$+ (**Q578** * $\mathbf{R_T}$) **5 + (0.2 *5) = 6**
23 CYCL DEF 272 OCM ROUGHING	
... **Q438 = -1 ;ROUGH-OUT TOOL** **...**	**-1**: The control assumes that the tool last used is 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.

...	
27 TOOL CALL Z "MILL_D6_FINISH"	
...	
29 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR** **...**	Resulting inside radius = R_T+ (**Q578** * R_T) **3 + (0.2 *3) = 3.6**
30 CYCL DEF 274 OCM FINISHING SIDE	
... **Q438 = -1 ;ROUGH-OUT TOOL** **...**	**-1**: The control assumes that the tool last used is the rough-out tool

Chamfering

- 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.

...	
33 TOOL CALL Z "NC_DEBURRING_D6"	
...	
35 CYCL DEF 277 OCM CHAMFERING	
... **QS438 = "MILL_D10_ROUGH" ;ROUGH-OUT TOOL** **...**	Rough-out tool of the last roughing operation

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:

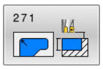
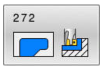
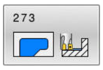
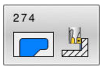
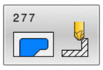
1. The control moves the tool to **Q260 CLEARANCE HEIGHT** at rapid traverse.
2. At **FMAX**, the tool moves to the starting point in the working plane and then to **Q200 SET-UP CLEARANCE**
3. The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**

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.

Overview

OCM cycles:

Soft key	Cycle	Page
	Cycle 271 OCM CONTOUR DATA (option 167) ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block	338
	Cycle 272 OCM ROUGHING (option 167) ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable	341
	Cycle 273 OCM FINISHING FLOOR (option 167) ■ Finishing with finishing allowance for the floor from Cycle **271** ■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances)	356
	Cycle 274 OCM FINISHING SIDE (option 167) ■ Finishing with side finishing allowance from Cycle **271**	360
	Cycle 277 OCM CHAMFERING (option 167) ■ Deburr the edges ■ Consider adjacent contours and walls	364

OCM standard figures:

Soft key	Cycle	Page
	Cycle 1271 OCM RECTANGLE (option 167) ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners	371
	Cycle 1272 OCM CIRCLE (option 167) ■ Definition of a circle ■ Input of the circle diameter	375
	Cycle 1273 OCM SLOT / RIDGE (option 167) ■ Definition of a slot or ridge ■ Input of the width and the length	378
	Cycle 1274 OCM CIRCULAR SLOT (option 167) ■ Definition of a circular slot ■ Input of width, pitch circle diameter and number of repetitions	382
	Cycle 1278 OCM POLYGON (option 167) ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners	386
	Cycle 1281 OCM RECTANGLE BOUNDARY (option 167) ■ Definition of a bounding rectangle	390
	Cycle 1282 OCM CIRCLE BOUNDARY (option #167) ■ Definition of a bounding circle	392

10.2 Cycle 271 OCM CONTOUR DATA (option 167)

ISO programming
G271

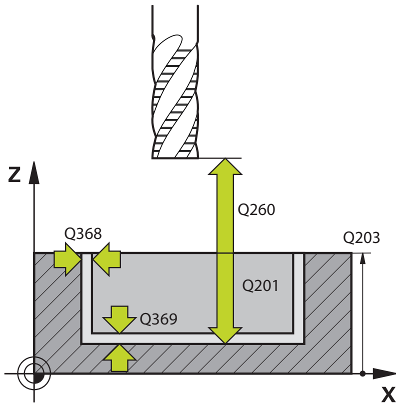
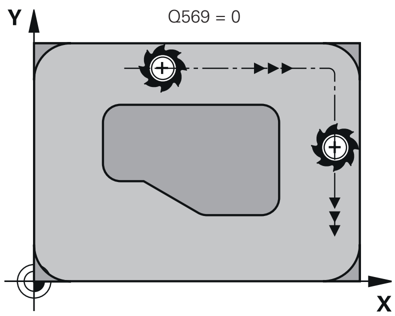
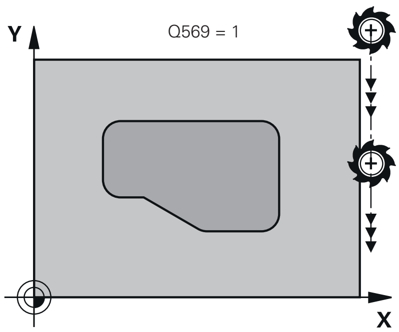
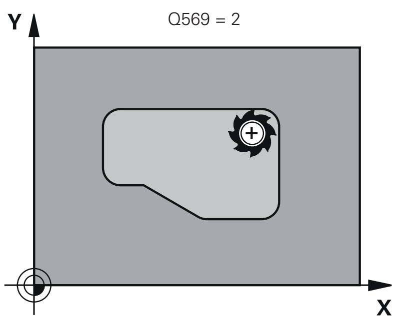
Application

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

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **271** are valid for Cycles **272** to **274**.

Cycle parameters

Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. 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 with **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              ;OPEN BOUNDARY
```

10.3 Cycle 272 OCM ROUGHING (option 167)

ISO programming

G272

Application

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

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

Further information: "OCM cutting data calculator (option 167)", Page 347

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 run

1. The tool uses positioning logic to move to the starting point
2. The control determines the starting point automatically based on the pre-positioning and the programmed contour

   Further information: "Positioning logic in OCM cycles", Page 336
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 342
   - **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

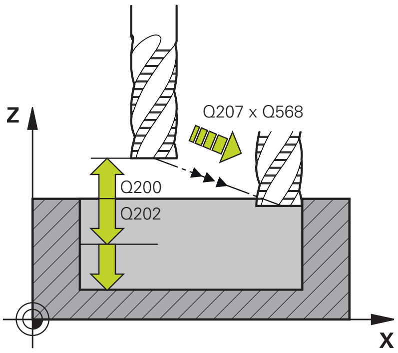
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH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

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

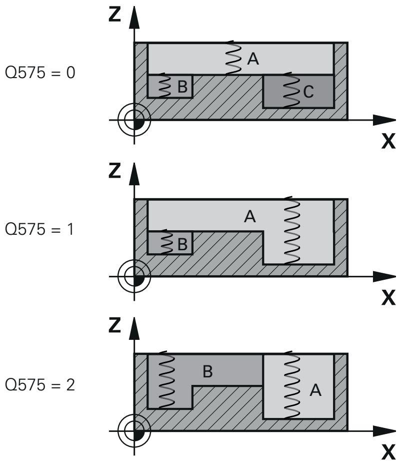
Notes on programming

- **CONTOUR DEF** / **SEL CONTOUR** will reset the tool radius that was used last. If you run this machining cycle with **Q438** = –1 after **CONTOUR DEF** / **SEL CONTOUR**, the control assumes that no pre-machining has taken place yet.
- If the path overlap factor **Q370 < 1**, a value of less than 1 is also recommended for the plunging factor **Q579** .
- 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**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used in Cycle **272** is the rough-out tool (default behavior) **0**: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0. Input: **-1...+32767.9** or max. **255** characters

Help graphic	Parameter
	Q577 Factor for appr./dept. radius? Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input: **0.15...0.99**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q576 Spindle speed? Spindle speed in revolutions per minute (rpm) for the roughing tool. **0**: The spindle speed from the **TOOL CALL** block will be used **> 0**: If a value greater than zero is entered, then this spindle speed will be used Input: **0...99999**
	Q579 Factor for plunging speed? Factor by which the control reduces the **SPINDLE SPEED Q576** for downfeed into the material. Input: **0.2...1.5**

Help graphic	Parameter
	**Q575 Infeed strategy (0/1)?** Type of downfeed: **0**: The control machines the contour from top to bottom **1**: The control machines the contour from bottom to top. The control does not always start with the deepest contour. The machining sequence is automatically calculated by the control. The total plunging path is often shorter than with strategy **2**. **2**: The control machines the contour from bottom to top. The control does not always start with the deepest contour. This strategy calculates the machining sequence such that the maximum length of the cutting edge is used. The resulting total plunging path is thus often larger than with strategy **1**. Depending on **Q568**, this may also result in a shorter machining time. Input: **0**, **1**, **2** The total plunging path is the sum of all plunging movements.

Example

11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+5	**;PLUNGING DEPTH ~**
Q370=+0.4	**;TOOL PATH OVERLAP ~**
Q207=+500	**;FEED RATE MILLING ~**
Q568=+0.6	**;PLUNGING FACTOR ~**
Q253=+750	**;F PRE-POSITIONING ~**
Q200=+2	**;SAFETY CLEARANCE ~**
Q438=-1	**;ROUGH-OUT TOOL ~**
Q577=+0.2	**;APPROACH RADIUS FACTOR ~**
Q351=+1	**;CLIMB OR UP-CUT ~**
Q576=+0	**;SPINDLE SPEED ~**
Q579=+1	**;PLUNGING FACTOR S ~**
Q575=+0	**;INFEED STRATEGY**

10.4 OCM cutting data calculator (option 167)

Fundamentals of the OCM cutting data calculator

Introduction

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

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

Requirements

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

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

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

Maintaining the cutting conditions

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

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

Chip removal

NOTICE

Caution: Danger to the tool and workpiece!

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

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

Process cooling

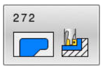
The OCM cutting data calculator recommends dry cutting with cooling by compressed air for most materials. The compressed air must be aimed directly at the cutting location. The best method is through the tool holder. If this is not possible, you can also mill with an internal coolant supply.

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

Operation

Opening the cutting data calculator

To open the cutting data calculator:



- Edit Cycle **272 OCM ROUGHING**

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

Closing the cutting data calculator

To close the cutting data calculator:

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

or

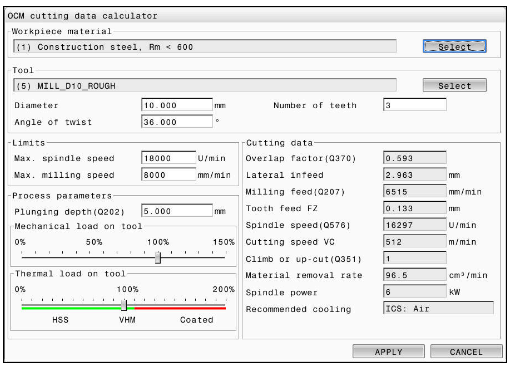
- Press the **END** or **CANCEL** soft key
- > The current entries are not stored.
- > The control does not apply any values to the cycle.

> The OCM cutting data calculator calculates associated values for these cycle parameters:
> - Plunging depth(Q202)
> - Overlap factor(Q370)
> - Spindle speed(Q576)
> - Climb or up-cut(Q351)
>
> When using the OCM cutting data calculator, you must not edit these parameters in the cycle later.

Fillable form

The control uses various colors in the fillable form:

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



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

Workpiece material

To select the workpiece material:

▶ Tap the **Select** button

> The control opens a selection list with various types of steel, aluminum, and titanium.

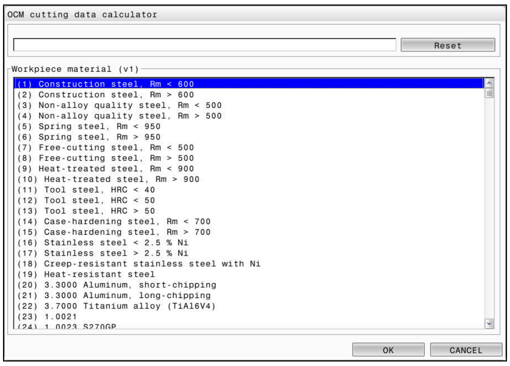
▶ Select the workpiece material

or

▶ Enter a search term in the search field

> The control displays the materials or material groups that were found. Press the **RESET** button to switch back to the original selection list.

▶ Apply your selection of the workpiece material with the **OK** button



Programming and operating notes:

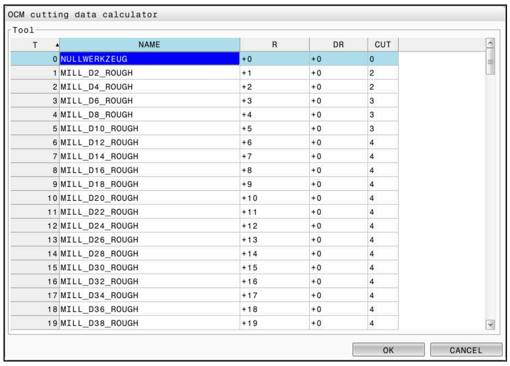
- If your material is not listed in the table, choose an appropriate material group or a material with similar cutting properties.
- The selection list also shows the version number of your current workpiece-material table. You can update this if necessary. You will find the workpiece-material table **ocm.xml** in the **TNC:\system_calcprocess** directory.

Tool

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

To select the tool:

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



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

Or proceed as follows without selecting a tool:

- Enter the Diameter
- Define the number of teeth
- Enter 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 and the number of teeth at any time. The modified value is **not** written to the tool table **tool.t**!
> - You will find the Angle of twist in the description of your tool, for example in the tool catalog of the tool manufacturer.

Limits

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

Input dialog	Description
Max. spindle speed	Maximum spindle speed in rpm permitted by the machine and the clamping situation: Input: **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 only effective in the range highlighted in green. This limiting depends on the maximum spindle speed, the maximum feed rate, and the selected material. ■ If the slider is in the red range, the control will use the maximum permissible value. Input: **0%...200%**

Further information: "Process parameters ", Page 354

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.

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 expectable spindle power, refer to the cutting data output.

> If the permissible spindle power is exceeded, you can first move the slider for the mechanical load to a lower value. If necessary, you can also reduce the plunging depth (a_p).
>
> Please note that at very high shaft speeds, a spindle running below its rated speed will not attain the rated power.
>
> If you wish to achieve a high material removal rate, you must ensure that chips are removed optimally.

Reduced load and low wear

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

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

Achieving an optimum result

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

Excessively high mechanical load

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

The following conditions are indications of excessive mechanical load:

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

Excessively high thermal load

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

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

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

Material removal rate is too low

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

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

Remedies for problems

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

Condition	Slider Mechanical load on tool	Slider Thermal load on tool	Miscellaneous
Vibrations (such as weak clamping or tools that project too far)	Decrease	Perhaps increase	Check the clamping
Undesired vibrations or chatter	Decrease	-	
Shaft of tool breaks	Decrease	-	Check the chip removal
Cutting edges of the tool break	Decrease	-	Check the chip removal
Excessive wear	Perhaps increase	Decrease	
The tool glows	Perhaps increase	Decrease	Check the cooling
Machining time is too long	Perhaps increase	Increase this first	
Excessive spindle load	Decrease	-	
Excessive axial force on spindle bearing	Decrease	-	■ Reduce the plunging depth ■ Use a tool with a lower angle of twist
Excessive radial force on spindle bearing	Decrease	-	

10.5 Cycle 273 OCM FINISHING FLOOR (option 167)

ISO programming
G273

Application

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

Requirements

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

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

Cycle run

1 The tool uses positioning logic to move to the starting point
  Further information: "Positioning logic in OCM cycles", Page 336
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!

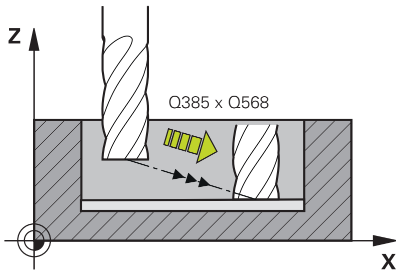
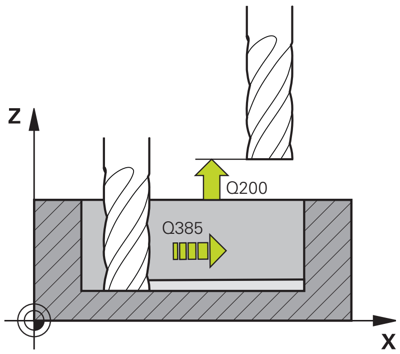
- Run a simulation to verify the machining sequence and the contour
- Use tools without a corner radius **R2** where possible

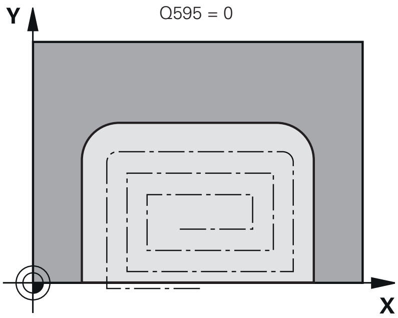
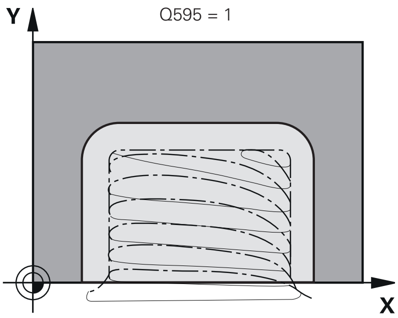
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.
- For finishing with Cycle **273**, the tool always works in climb milling mode.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Note on programming

- If you use an overlap factor greater than 1, residual material may be left over. Check the contour using the program verification graphics and slightly change the overlap factor, if necessary. This allows another distribution of cuts, which often provides the desired results.

Cycle parameters

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = lateral infeed k. The overlap is considered to be the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input: **0.0001...1.9999** or **PREDEF**
	**Q385 Finishing feed rate?** Traversing speed of the tool in mm/min for floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q568 Factor for plunging feed rate? Factor by which the control reduces the feed rate **Q385** for downfeed into the material. Input: **0.1...1**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	**Q200 Set-up clearance?** Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters

Help graphic	Parameter
	**Q595 Strategy (0/1)?** Machining strategy for finishing **0**: Equidistant strategy = constant distances between paths **1**: Strategy with constant contact angle Input: **0**, **1**
	**Q577 Factor for appr./dept. radius?** Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input: **0.15...0.99**

Example

```
11 CYCL DEF 273 OCM FINISHING FLOOR ~
    Q370=+1              ;TOOL PATH OVERLAP ~
    Q385=+500            ;FINISHING FEED RATE ~
    Q568=+0.3            ;PLUNGING FACTOR ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q438=-1              ;ROUGH-OUT TOOL ~
    Q595=+1              ;STRATEGY ~
    Q577=+0.2            ;APPROACH RADIUS FACTOR
```

10.6 Cycle 274 OCM FINISHING SIDE (option 167)

ISO programming
G274

Application

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

Cycle **274** can also be used for contour milling.

Proceed as follows:

- Define the contour to be milled as a single island (without pocket boundary)
- Enter the finishing allowance (**Q368**) in Cycle **271** to be greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Requirements

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

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

Cycle run

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 336
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

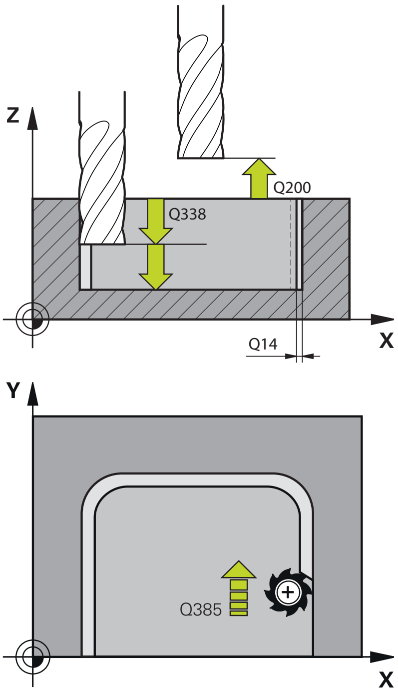
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour and the allowance programmed in Cycle **271**.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- You can execute this cycle using a grinding tool.
- 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: User's Manual for **Klartext Programming**

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 soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Example

```
11 CYCL DEF 274 OCM FINISHING SIDE ~
    Q338=+0              ;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 OR UP-CUT
```

10.7 Cycle 277 OCM CHAMFERING (option 167)

ISO programming
G277

Application

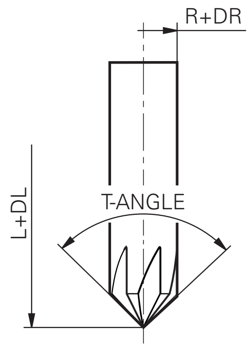
Cycle **277 OCM CHAMFERING** enables you to deburr edges of complex contours that you roughed out using OCM cycles.

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

Requirements

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

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



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

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

Cycle run

1. The tool uses positioning logic to move to the starting point. This point is determined automatically based on the programmed contour

   Further information: "Positioning logic in OCM cycles", Page 336
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). For machining the chamfer, the tool uses the milling feed rate **Q207**
5. Then, the tool is retracted from the contour in a tangential or vertical movement (depending on the available space).
6. If there are several contours, the control positions the tool at clearance height after each contour and then moves it to the next starting point. Steps 3 to 6 are repeated until the programmed contour is completely chamfered
7. 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 only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for chamfering. The starting point depends on the available space.
- 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.

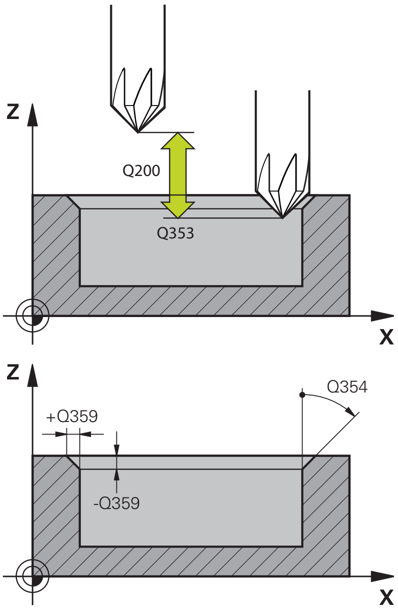
  Further information: User's Manual for **Klartext Programming**
- If the roughing operations have not completely removed the material before chamfering, you need to define the last roughing tool in **QS438 ROUGH-OUT TOOL**, in order to prevent damage to the contour.

  Further information: "Procedure regarding residual material in inside corners", Page 333

Note on programming

- If the value of parameter **Q353 DEPTH OF TOOL TIP** is less than the value of parameter **Q359 CHAMFER WIDTH**, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q353 Depth of tool tip?** Distance between theoretical tool tip and workpiece surface coordinate. This value has an incremental effect. Input: **-999.9999...-0.0001**
	Q359 Width of chamfer (-/+)? Width or depth of chamfer: **-**: Depth of chamfer **+**: Width of chamfer This value has an incremental effect. Input: **-999.9999...+999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for positioning Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via soft key. In addition, you can enter the tool name via the **Tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Help graphic	Parameter
	Q354 Angle of chamfer? Angle of the chamfer **0**: The chamfer angle is half the defined **T-ANGLE** from the tool table **> 0**: The chamfer angle is compared to the value of **T-ANGLE** from the tool table. If these two values do not match, the control will display an error message. Input: **0...89**

Example

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

10.8 OCM standard figures

Fundamentals

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

The cycles offer the following advantages:

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

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

The control offers the following cycles for standard figures:

- **1271 OCM RECTANGLE**, see Page 371
- **1272 OCM CIRCLE**, see Page 375
- **1273 OCM SLOT / RIDGE**, see Page 378
- **1274 OCM CIRCULAR SLOT**, see Page 382
- **1278 OCM POLYGON**, see Page 386

The control provides the following cycles for figure boundaries:

- **1281 OCM RECTANGLE BOUNDARY**, see Page 390
- **1282 OCM CIRCLE BOUNDARY**, see Page 392

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
- Press the **ENTER TEXT** soft key
- 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.

10.9 Cycle 1271 OCM RECTANGLE (option 167)

ISO programming
G1271

Application

Use the figure cycle **1271 OCM RECTANGLE** to program a rectangle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program tolerances for the lengths.

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

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

Notes

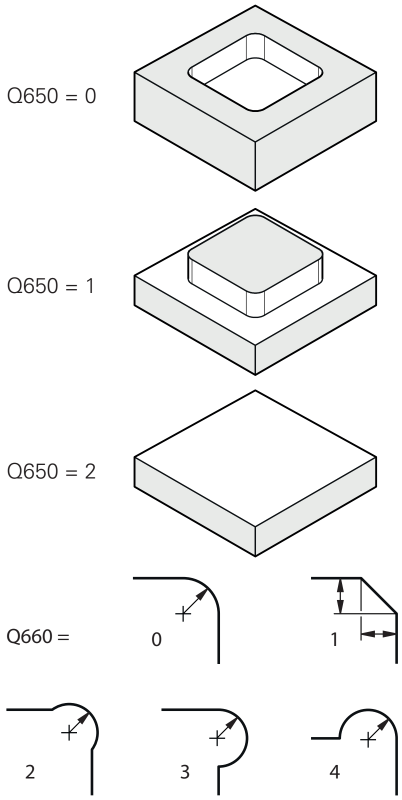
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1271** are valid for the OCM machining cycles **272** to **274** and **277**.

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 369

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 369

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

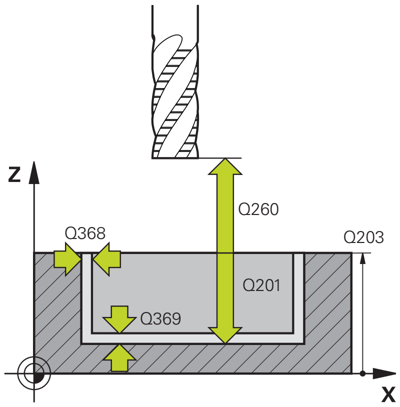
4: Tool position = Upper left corner

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 with **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              ;FIGURE TYPE ~
    Q218=+60             ;FIRST SIDE LENGTH ~
    Q219=+40             ;2ND SIDE LENGTH ~
    Q660=+0              ;CORNER TYPE ~
    Q220=+0              ;CORNER RADIUS ~
    Q367=+0              ;POCKET POSITION ~
    Q224=+0              ;ANGLE OF ROTATION ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q201=-10             ;DEPTH ~
    Q368=+0              ;ALLOWANCE FOR SIDE ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q260=+50             ;CLEARANCE HEIGHT ~
    Q578=+0.2            ;INSIDE CORNER FACTOR
```

10.10 Cycle 1272 OCM CIRCLE (option 167)

ISO programming
G1272

Application

Use figure cycle **1272 OCM CIRCLE** to program a circle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the diameter.

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

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

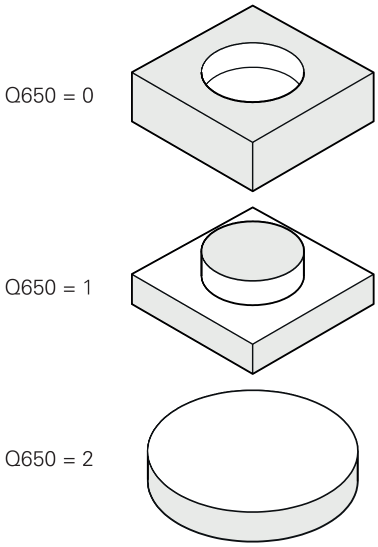
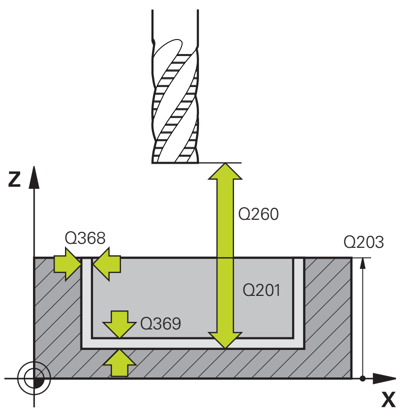
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1272** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1272** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- 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 369 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 with **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            ;FIGURE TYPE ~
    Q223=+50           ;CIRCLE DIAMETER ~
    Q367=+0            ;POCKET POSITION ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q201=-20           ;DEPTH ~
    Q368=+0            ;ALLOWANCE FOR SIDE ~
    Q369=+0            ;ALLOWANCE FOR FLOOR ~
    Q260=+100          ;CLEARANCE HEIGHT ~
    Q578=+0.2          ;INSIDE CORNER FACTOR
```

10.11 Cycle 1273 OCM SLOT / RIDGE (option 167)

ISO programming
G1273

Application

Use figure cycle **1273 OCM SLOT / RIDGE** to program a slot or a ridge. This figure cycle also allows you to program a boundary for face milling. In addition, you can program a tolerance for the width and the length.

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

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

Notes

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

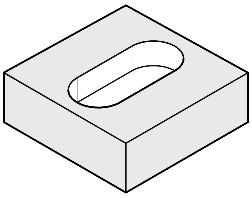
Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- 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.

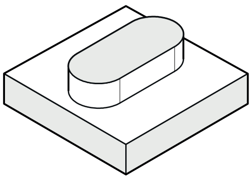
Cycle parameters

Help graphic

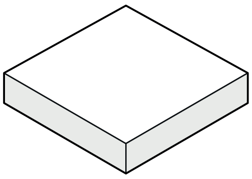
Q650 = 0



Q650 = 1



Q650 = 2



Parameter

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 369

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 369

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

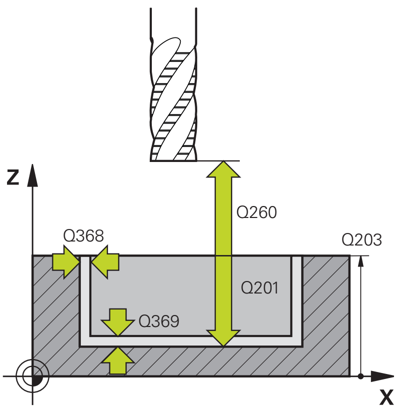
4: Tool position = Right end of figure

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 with **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            ;FIGURE TYPE ~
    Q219=+10           ;SLOT WIDTH ~
    Q218=+60           ;SLOT LENGTH ~
    Q367=+0            ;SLOT POSITION ~
    Q224=+0            ;ANGLE OF ROTATION ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q201=-20           ;DEPTH ~
    Q368=+0            ;ALLOWANCE FOR SIDE ~
    Q369=+0            ;ALLOWANCE FOR FLOOR ~
    Q260=+100          ;CLEARANCE HEIGHT ~
    Q578=+0.2          ;INSIDE CORNER FACTOR
```

10.12 Cycle 1274 OCM CIRCULAR SLOT (option 167)

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

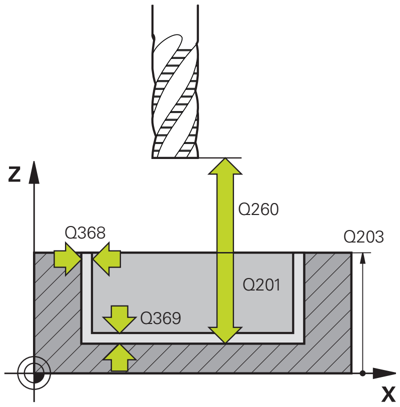
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1274** is DEF-active, which means that Cycle **1274** becomes active as soon as it has been defined in the NC program.
- The machining data defined in Cycle **1274** are valid for the OCM machining cycles **272** 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 369 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**
	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 left figure arc **2**: Tool position = center of figure center axis **3**: Tool position = center of right figure arc Input: **0**, **1**, **2**, **3**

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 with **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
```

10.13 Cycle 1278 OCM POLYGON (option 167)

ISO programming
G1278

Application

Use figure cycle **1278 OCM POLYGON** to program a polygon. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the reference diameter.

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

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

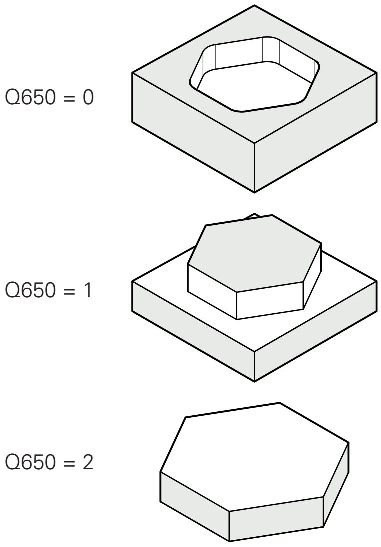
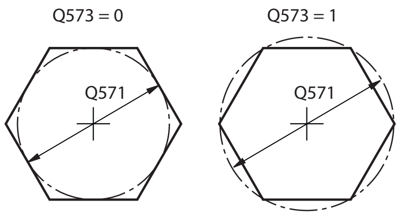
Notes

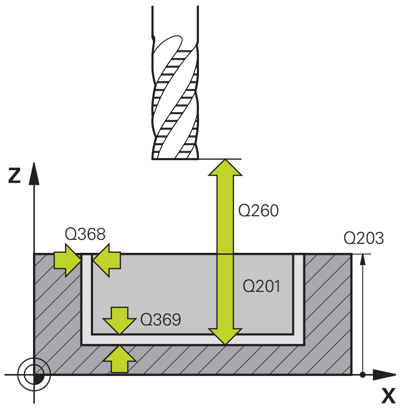
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1278** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1278** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- 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**
	**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 369 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**

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

10.14 Cycle 1281 OCM RECTANGLE BOUNDARY (option 167)

ISO programming
G1281

Application

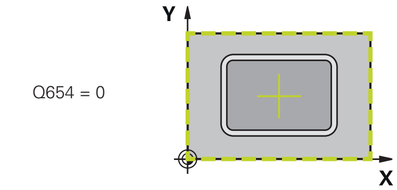
Use Cycle **1281 OCM RECTANGLE BOUNDARY** to program a rectangular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

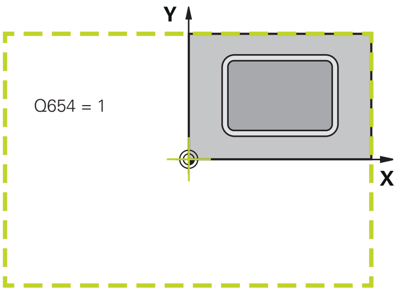
Notes

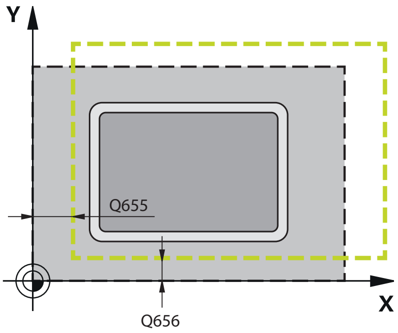
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1281** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1281** are valid for Cycles **1271** to **1274** and **1278**.

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           ;POSITION REFERENCE ~
    Q655=+0           ;SHIFT 1 ~
    Q656=+0           ;SHIFT 2
```

10.15 Cycle 1282 OCM CIRCLE BOUNDARY (option #167)

ISO programming
G1282

Application

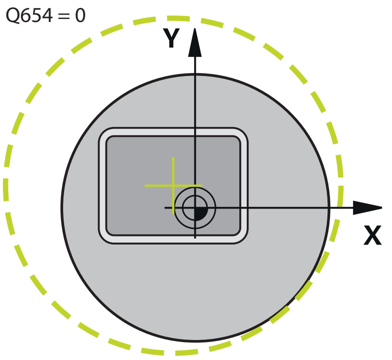
Cycle **1282 OCM CIRCLE BOUNDARY** allows you to program a circular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

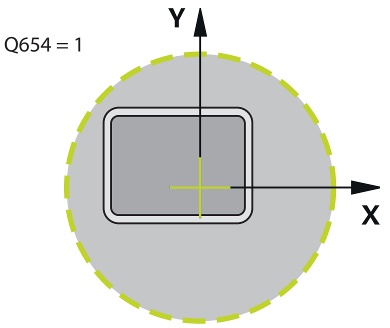
Notes

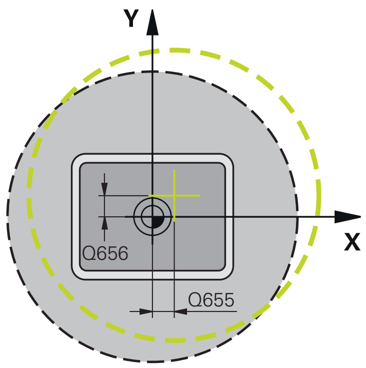
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1282** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1282** are valid for Cycles **1271** to **1274** and **1278**.

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**

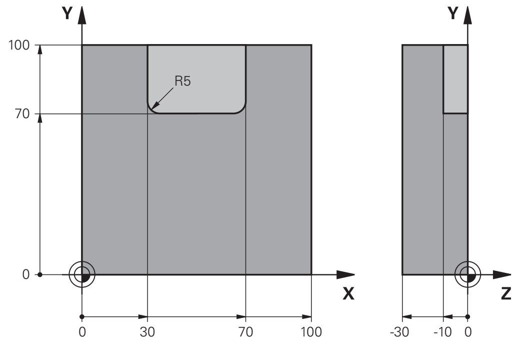
Example

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

10.16 Programming examples

Example: Open pocket and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will program an open pocket that is defined by means of an island and a boundary. Machining includes roughing and finishing of an open pocket.



Program sequence

- Tool call: Roughing cutter (Ø 20 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Roughing cutter (Ø 8 mm)
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 6 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**

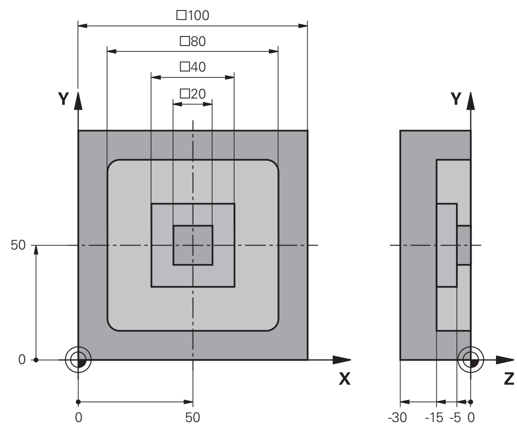
0 BEGIN PGM OCM_POCKET MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S8000 F1500	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+1 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+6500 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;INFEED STRATEGY	

8 CYCL CALL	; Cycle call
9 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+10 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;INFEED STRATEGY	
12 CYCL CALL	; Cycle call
13 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
14 L Z+100 R0 FMAX M3	
15 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
16 CYCL CALL	; Cycle call
17 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;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 OR UP-CUT	
18 CYCL CALL	; Cycle call
19 M30	; End of program
20 LBL 1	; Contour subprogram 1
21 L X+0 Y+0	
22 L X+100	
23 L Y+100	

24 L X+0	
25 L Y+0	
26 LBL 0	
27 LBL 2	; Contour subprogram 2
28 L X+0 Y+0	
29 L X+100	
30 L Y+100	
31 L X+70	
32 L Y+70	
33 RND R5	
34 L X+30	
35 RND R5	
36 L Y+100	
37 L X+0	
38 L Y+0	
39 LBL 0	
40 END PGM OCM_POCKET MM	

Example: Program various depths with OCM cycles

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



Program sequence

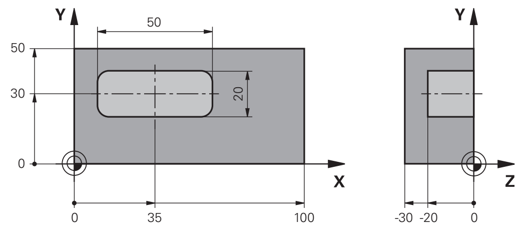
- Tool call: Roughing cutter (Ø 10 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 6 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**

0 BEGIN PGM OCM_DEPTH MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-30	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 5 Z S8000 F1500	; Tool call (diameter: 10 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 I3 = LBL 3 DEPTH5	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-15 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+0 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 CYCL CALL	; Cycle call
9 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 273 OCM FINISHING FLOOR ~	

Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 CYCL CALL	; Cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;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 OR UP-CUT	
14 CYCL CALL	; Cycle call
15 M30	; End of program
16 LBL 1	; Contour subprogram 1
17 L X-40 Y-40	
18 L X+40	
19 L Y+40	
20 L X-40	
21 L Y-40	
22 LBL 0	
23 LBL 2	; Contour subprogram 2
24 L X-10 Y-10	
25 L X+10	
26 L Y+10	
27 L X-10	
28 L Y-10	
29 LBL 0	
30 LBL 3	; Contour subprogram 3
31 L X-20 Y-20	
32 L 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.



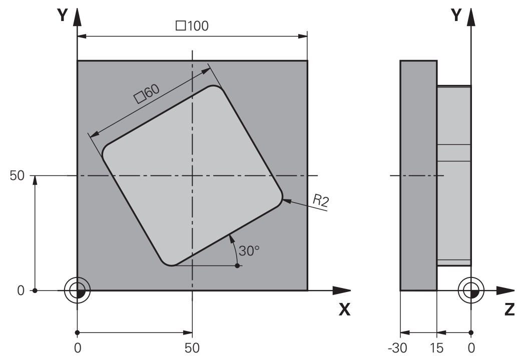
Program sequence

- 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 OR UP-CUT ~	
Q576=+8000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 L X+0 Y+0 R0 FMAX M99	; Cycle call
9 TOOL CALL 4 Z S6000 F4000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 272 OCM ROUGHING ~	

Q202=+25 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+6 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
12 L X+0 Y+0 R0 FMAX M99	; Cycle call
13 M30	; End of program
14 LBL 1	; Contour subprogram 1
15 L X+0 Y+0	
16 L Y+50	
17 L X+100	
18 L Y+0	
19 L X+0	
20 LBL 0	
21 LBL 2	; Contour subprogram 2
22 L X+10 Y+30	
23 L Y+40	
24 RND R5	
25 L X+60	
26 RND R5	
27 L Y+20	
28 RND R5	
29 L X+10	
30 RND R5	
31 L Y+30	
32 LBL 0	
33 END PGM FACE_MILL MM	

Example: Contour with OCM figure cycles



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

Program sequence

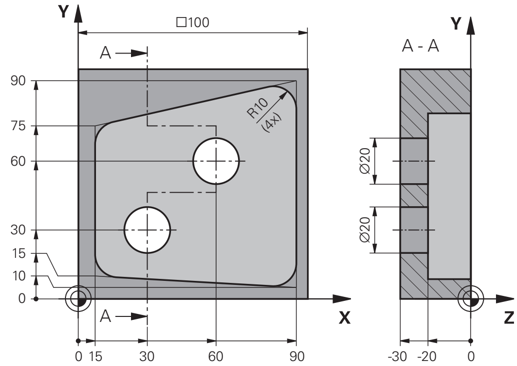
- Tool call: Roughing cutter (Ø 8 mm)
- Define Cycle **1271**
- Define Cycle **1281**
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 8 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**

0 BEGIN PGM OCM_FIGURE MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 1271 OCM RECTANGLE ~	
Q650=+1 ;FIGURE TYPE ~	
Q218=+60 ;FIRST SIDE LENGTH ~	
Q219=+60 ;2ND SIDE LENGTH ~	
Q660=+0 ;CORNER TYPE ~	
Q220=+2 ;CORNER RADIUS ~	
Q367=+0 ;POCKET POSITION ~	
Q224=+30 ;ANGLE OF ROTATION ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR	
6 CYCL DEF 1281 OCM RECTANGLE BOUNDARY ~	
Q651=+100 ;LENGTH 1 ~	
Q652=+100 ;LENGTH 2 ~	
Q654=+0 ;POSITION REFERENCE ~	
Q655=+0 ;SHIFT 1 ~	
Q656=+0 ;SHIFT 2	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.4 ;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 OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
9 TOOL CALL 24 Z S10000 F2000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+15 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
14 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
15 M30	; End of program
16 END PGM OCM_FIGURE MM	

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 ;DEPTH REFERENCE	
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 ;OPEN BOUNDARY	
12 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.441 ;TOOL PATH OVERLAP ~	
Q207=+6000 ;FEED RATE MILLING ~	

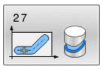
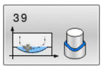
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+13626 ;SPINDLE SPEED ~	
Q579=+1 ;PLUNGING FACTOR S ~	
Q575=+2 ;INFEED STRATEGY	
13 CYCL CALL	
14 M30	; End of program
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	

11

Cycles: Cylinder Surface

11.1 Fundamentals

Overview of cylindrical surface cycles

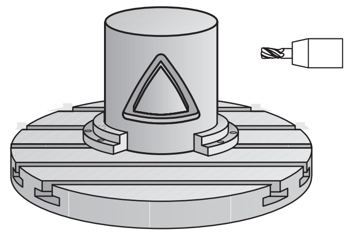
Soft key	Cycle	Page
	Cycle 27 CYLINDER SURFACE (option 8) ■ Milling of guide slots on the cylinder surface ■ Slot width is equal to tool radius	407
	Cycle 28 CYLINDRICAL SURFACE SLOT (option 8) ■ Milling of guide slots on the cylinder surface ■ Input of the slot width	410
	Cycle 29 CYL SURFACE RIDGE (option 8) ■ Milling of a ridge on the cylinder surface ■ Input of the ridge width	415
	Cycle 39 CYL. SURFACE CONTOUR (option 8) ■ Milling of a contour on the cylinder surface	419

11.2 Cycle 27 CYLINDER SURFACE (option 8)

ISO programming

G127

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to program a contour in two dimensions and then transfer it onto a cylindrical surface. Use Cycle **28** to mill guide slots on the cylinder.

Describe the contour in a subprogram that you program with Cycle **14 CONTOUR**.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

The 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

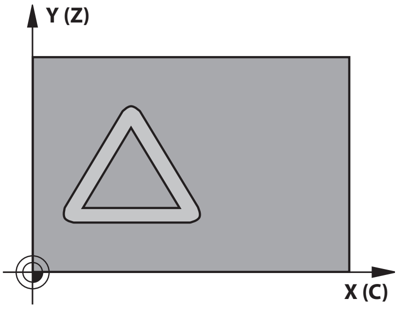
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.
- This cycle can also be used in a tilted working plane.

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

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**

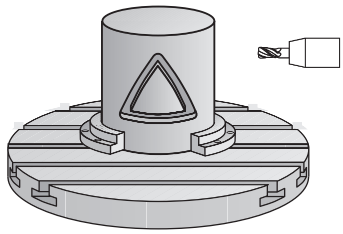
Example

```
11 CYCL DEF 27 CYLINDER SURFACE ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q6=+0                   ;SET-UP CLEARANCE ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q16=+0                  ;RADIUS ~
    Q17=+0                  ;TYPE OF DIMENSION
```

11.3 Cycle 28 CYLINDRICAL SURFACE SLOT (option 8)

ISO programming
G128

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

With this cycle you can program a guide slot in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle **27**, with this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel walls by using a tool that is exactly as wide as the slot.

The smaller the tool is with respect to the slot width, the larger the distortion in circular arcs and oblique line segments. To minimize this process-related distortion, you can define the parameter **Q21**. This parameter specifies the tolerance with which the control machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.

Program the center path of the contour together with the tool radius compensation. With the radius compensation you specify whether the control cuts the slot with climb milling or up-cut milling.

Cycle run

1. The control positions the tool above the infeed point.
2. The control moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate **Q12**. The approaching behavior depends on the **ConfigDatum CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004) parameter
3. At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate **Q12** while respecting the finishing allowance for the side
4. At the end of the contour, the control moves the tool to the opposite slot wall and returns to the infeed point.
5. Steps 2 to 3 are repeated until the programmed milling depth **Q1** is reached.
6. If you defined the tolerance in **Q21**, the control then re-machines the slot walls to be as parallel as possible
7. Finally, the tool retracts in the tool axis to the clearance height.

> 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, check the end position of the tool after the cycle
- After the cycle, program absolute coordinates (no incremental coordinates)

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.
- This cycle can also be used in a tilted working plane.

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

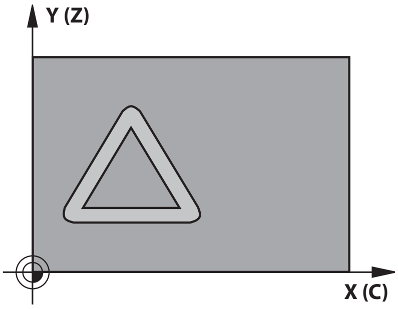
Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**
	Q20 Slot width? Width of the slot to be machined Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q21 Tolerance? If you use a tool smaller than the programmed slot width **Q20**, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance **Q21**, the control adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With **Q21**, you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the re-machining takes. **Recommendation**: Use a tolerance of 0.02 mm. **Function inactive**: Enter 0 (default setting). Input: **0...9.9999**

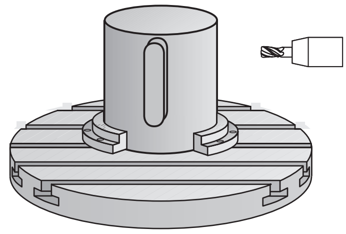
Example

```
11 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~
    Q1=-20              ;MILLING DEPTH ~
    Q3=+0               ;ALLOWANCE FOR SIDE ~
    Q6=+2               ;SET-UP CLEARANCE ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q16=+0              ;RADIUS ~
    Q17=+0              ;TYPE OF DIMENSION ~
    Q20=+0              ;SLOT WIDTH ~
    Q21=+0              ;TOLERANCE
```

11.4 Cycle 29 CYL SURFACE RIDGE (option 8)

ISO programming
G129

Application

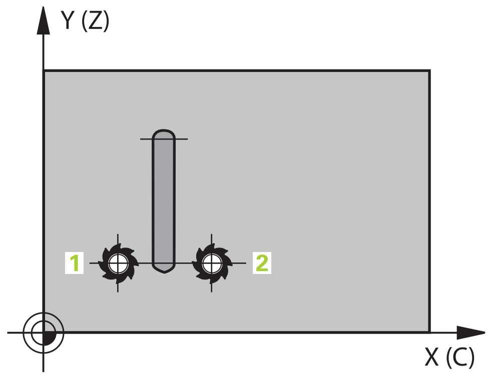


Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle, the control adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center path of the ridge together with the tool radius compensation. With the radius compensation you specify whether the control cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge, the control will always add a semi-circle whose radius corresponds to half the ridge width.

Cycle sequence



1 The control positions the tool above the starting point of machining. The control calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins to the left (**1**, RL = climb milling) or to the right of the ridge (**2**, RR = up-cut milling).
2 After the control has positioned the tool to the first plunging depth, the tool moves on a circular arc at the milling feed rate **Q12** tangentially to the ridge wall. A finishing allowance programmed for the side is taken into account.
3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate **Q12** until the ridge is completed.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth **Q1** is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.

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

Notes

This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	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**
	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

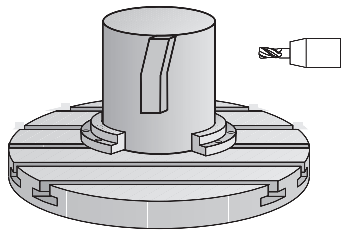
```
11 CYCL DEF 29 CYL SURFACE RIDGE ~
    Q1=-20              ;MILLING DEPTH ~
    Q3=+0               ;ALLOWANCE FOR SIDE ~
    Q6=+2               ;SET-UP CLEARANCE ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q16=+0              ;RADIUS ~
    Q17=+0              ;TYPE OF DIMENSION ~
    Q20=+0              ;RIDGE WIDTH
```

11.5 Cycle 39 CYL. SURFACE CONTOUR (option 8)

ISO programming

G139

Application



Refer to your machine manual.

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

This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the milled contour are always parallel to the cylinder axis.

Describe the contour in a subprogram that you program with Cycle **14 CONTOUR**.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

Unlike in Cycles **28** and **29**, in the contour subprogram, you define the contour actually to be machined.

Cycle sequence

1. The control positions the tool above the starting point of machining. The control locates the starting point next to the first point defined in the contour subprogram offset by the tool diameter
2. The control then moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate **Q12**. A finishing allowance programmed for the side is taken into account. The approach behavior depends on the machine parameter **apprDepCylWall** (no. 201004)
3. At the first plunging depth, the tool mills along the programmed contour at the milling feed rate **Q12** until the contour train is complete.
4. The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5. Steps 2 to 4 are repeated until the programmed milling depth **Q1** is reached.
6. Finally, the tool retracts in the tool axis to the clearance height.

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

Notes

This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

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

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line

Cycle parameters

Help graphic	Parameter
	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**

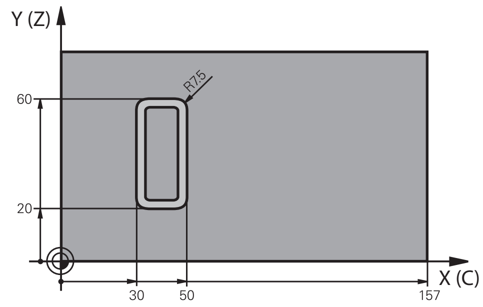
Example

```
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              ;TYPE OF DIMENSION
```

11.6 Programming examples

Example: Cylinder surface with Cycle 27

- Machine with B head and C table
- Cylinder centered on rotary table
- Preset is on the underside, in the center of the rotary table

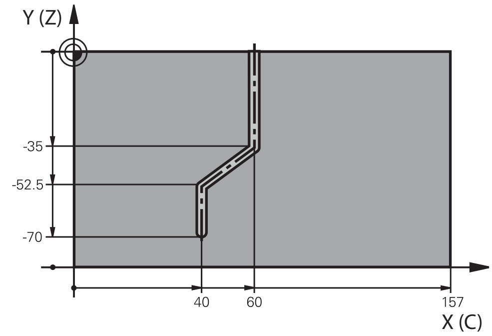


0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call (diameter: 7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-7 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q10=-4 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+250 ;FEED RATE F. ROUGHNG ~	
Q16=+25 ;RADIUS ~	
Q17=+1 ;TYPE OF DIMENSION	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram
13 L X+40 Y-20 RL	; Rotary axis data in mm (Q17 = 1)
14 L X+50	
15 RND R7.5	
16 L Y-60	
17 RND R7.5	
18 L IX-20	
19 RND R7.5	
20 L Y-20	
21 RND R7.5	
22 L X+40 Y-20	

23 LBL 0	
24 END PGM 5 MM	

Example: Cylinder surface with Cycle 28

- Cylinder centered on rotary table
- Machine with B head and C table
- Preset is at the center of the rotary table
- Description of the path of the tool center in the contour subprogram



0 BEGIN PGM 4 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call, tool axis (Z), diameter (7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR 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 ;TYPE OF DIMENSION ~	
Q20=+10 ;SLOT WIDTH ~	
Q21=+0.02 ;TOLERANCE	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram, description of the path of the tool center
13 L X+60 Y+0 RL	; Rotary axis data in mm (Q17 = 1)
14 L Y-35	
15 L X+40 Y-52.5	
16 L X-70	
17 LBL 0	
18 END PGM 4 MM	

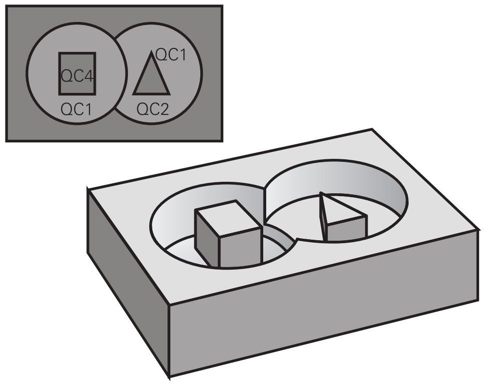
12

Cycles: Contour Pocket with Contour Formula

12.1 SL or OCM cycles with complex contour formula

Fundamentals

Using complex contour formulas, you can combine several subcontours (pockets or islands) to program complex contours. You define the individual subcontours (geometry data) in separate NC programs 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.



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
...
```

Selecting an NC program with contour definitions

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

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

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

SEL CONTOUR

- Press the **SEL CONTOUR** soft key.

You can enter contours in the following ways:

Soft key	Function
CONTOUR <FILE> SELECT FILE	Define the name of the contour or Press the **SELECT FILE** soft key
CONTOUR <FILE>=QS	Define the number of a QS parameter
CONTOUR LBL NR	Define the number of a label
CONTOUR LBL NAME	Define the name of the label
CONTOUR LBL QS	Define the number of a QS parameter for a label

Programming notes:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
- Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR** is no longer necessary if you use **SEL CONTOUR**.

Defining contour descriptions

Using the **DECLARE CONTOUR** function in your NC program, you enter the path for NC programs from which the control extracts the contour descriptions. In addition, you can select a separate depth for this contour description.

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

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

DECLARE CONTOUR

- Press the **DECLARE CONTOUR** soft key.
- Enter the number for the contour designator **QC**
- Press the **ENT** key
- Enter the full name of the NC program with the contour descriptions and confirm with the **ENT** key.

  or

SELECT FILE

- Press the **SELECT FILE** soft key and select the desired NC program
- Define a separate depth for the selected contour
- Press the **END** key

> Programming notes:
> - If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
> - With the entered contour designators **QC** you can include the various contours in the contour formula.
> - If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).
> - The control will only take different depths (**DEPTH**) into account if the elements overlap. In case of pure islands inside a pocket, this is not the case. Use a simple contour formula for this purpose.
>   **Further information:** "SL or OCM cycles with simple contour formula", Page 437

Entering a complex contour formula

You can use soft keys to interlink various contours in a mathematical formula.

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

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

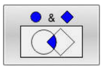
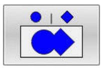
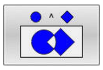
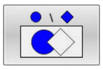
CONTOUR FORMULA

- Press the **CONTOUR FORMULA** soft key
- Enter the number for the contour designator **QC**

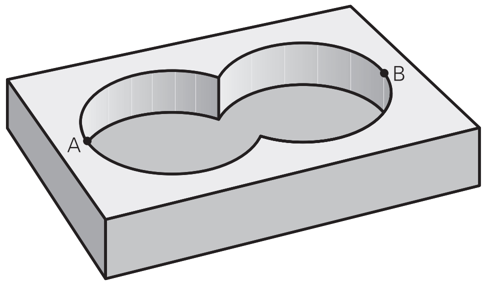
ENT

- Press the **ENT** key

The control displays the following soft keys:

Soft key	Mathematical function
	**Intersected with** (e.g., **QC10= QC1 & QC5**)
	**Joined with** (e.g., **QC25= QC7| QC18**)
	**joined with, but without intersection** (e.g., **QC12 = QC5 ^ QC25**)
	**without** (e.g., **QC25 = QC1 \ QC2**)
(	**Open parenthesis** (e.g., **QC12 = QC1 & (QC2 | QC3)**)
)	**Close parenthesis** (e.g., **QC12 = QC1 & (QC2 | QC3)**)
	Define single contour (e.g., **QC12 = QC1**)

Superimposed contours



By default, the control considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: overlapping pockets

> The following examples are contour description programs that are defined in a contour definition program. The contour definition program is called through the **SEL CONTOUR** function in the actual main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Contour description program 1: pocket A

```
0 BEGIN PGM POCKET MM
1 L X+10 Y+50 R0
2 CC X+35 Y+50
3 C X+10 Y+50 DR-
4 END PGM POCKET MM
```

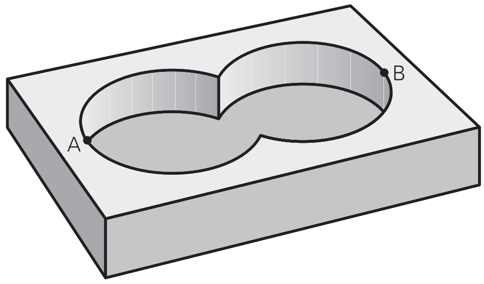
Contour description program 2: pocket B

```
0 BEGIN PGM POCKET2 MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET2 MM
```

Area of inclusion

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

- Areas A and B must have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.



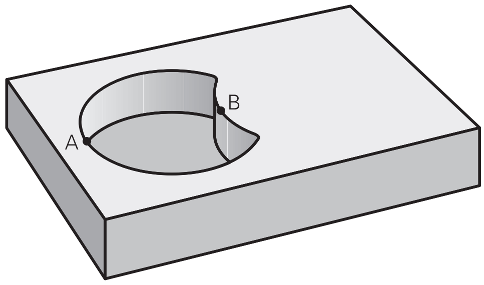
Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 | QC2
* - ...
```

Area of exclusion

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

- Surfaces A and B must be 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.



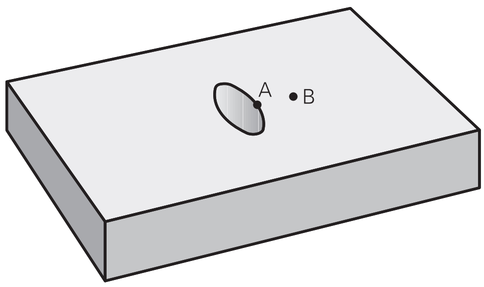
Contour definition program:

```
* - ...
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 be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "intersection with" function.



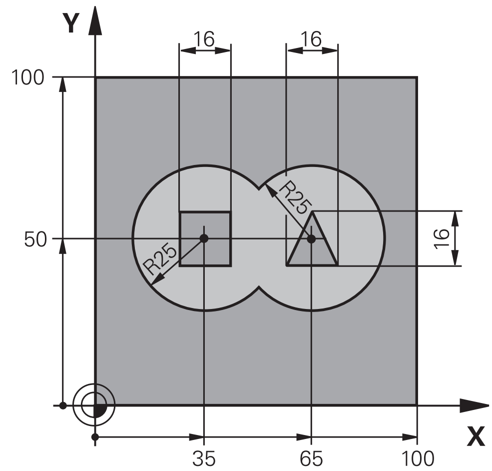
Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 & QC2
* - ...
```

Machining contours with SL or OCM cycles

> The entire contour is machined with the SL cycles (see "Overview", Page 282) or the OCM cycles (see "Overview", Page 337).

Example: Roughing and finishing superimposed contours with the contour formula



0 BEGIN PGM CONTOUR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	; Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S2500	; Tool call: roughing cutter
4 L Z+250 R0 FMAX M3	; Retract the tool
5 SEL CONTOUR "MODEL"	; Specify the contour definition program
6 CYCL DEF 20 CONTOUR DATA ~	; Define the general machining parameters
Q1=-20 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0.5 ;ALLOWANCE FOR SIDE ~	
Q4=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+100 ;CLEARANCE HEIGHT ~	
Q8=+0.1 ;ROUNDING RADIUS ~	
Q9=-1 ;ROTATIONAL DIRECTION	
7 CYCL DEF 22 ROUGH-OUT ~	; Cycle definition: rough-out
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+350 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+150 ;FEED RATE FOR RECIP. ~	
Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+100 ;FEED RATE FACTOR ~	

Q404=+0 ;FINE ROUGH STRATEGY	
8 CYCL CALL	; Cycle call: rough-out
9 TOOL CALL 23 Z S5000	; Tool call: finishing cutter
10 L Z+250 R0 FMAX M3	
11 CYCL DEF 23 FLOOR FINISHING ~	; Cycle definition: floor finishing
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+200 ;FEED RATE F. ROUGHNG ~	
Q208=+99999 ;RETRACTION FEED RATE	
12 CYCL CALL	; Cycle call: floor finishing
13 CYCL DEF 24 SIDE FINISHING ~	; Cycle definition: side finishing
Q9=+1 ;ROTATIONAL DIRECTION ~	
Q10=-10 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+400 ;FEED RATE F. ROUGHNG ~	
Q14=+0 ;ALLOWANCE FOR SIDE	
14 CYCL CALL	; Cycle call: side finishing
15 L Z+250 R0 FMAX	; Retract the tool, end program
16 M30	
17 END PGM CONTOUR MM	

Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	
1 DECLARE CONTOUR QC1 = "120"	; Define the contour label for the NC program "120"
2 Q1 = 35	; Assign the values for parameters used in PGM "121"
3 Q2 = 50	
4 Q3 = 25	
5 DECLARE CONTOUR QC2 = "121"	; Define the contour label for the NC program "121"
6 DECLARE CONTOUR QC3 = "122"	; Define the contour label for the NC program "122"
7 DECLARE CONTOUR QC4 = "123"	; Define the contour label for the NC program "123"
8 QC10 = (QC1 | QC2) \ QC3 \ QC4	; Contour formula
9 END PGM MODEL MM	

Contour description program for circle at right:

0 BEGIN PGM 120 MM	
1 CC X+65 Y+50	
2 LP PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM 120 MM	

Contour description program for circle at left:

0 BEGIN PGM 121 MM	
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM 121 MM	

Contour description program for triangle at right:

0 BEGIN PGM 122 MM	
1 L X+73 Y+42 R0	
2 L X+65 Y+58	
3 L X+58 Y+42	
4 L X+73	
5 END PGM 122 MM	

Contour description program for square at left:

0 BEGIN PGM 123 MM	
1 L X+27 Y+58 R0	
2 L X+43	
3 L Y+42	
4 L X+27	
5 L Y+58	
6 END PGM 123 MM	

12.2 SL or OCM cycles with simple contour formula

Fundamentals

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.

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 set-up clearance, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM CONTOUR DATA**.

Entering a simple contour formula

You can use soft keys to interlink various contours in a mathematical formula.

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

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

CONTOUR DEF

- Press the **CONTOUR DEF** soft key
- Press the **ENT** key
- > The control opens the dialog for entering the contour formula.
- Enter the first subcontour **P1**. Confirm with the **ENT** key

POCKET

- Press the **POCKET (P)** soft key

  or

ISLAND

- Press the **ISLAND (I)** soft key
- Enter the second subcontour and confirm with the **ENT** key
- If needed, enter the depth of the second subcontour. Press the **ENT** key
- > Carry on with the dialog as described above until you have entered all subcontours.
- Define void areas **V** as needed

> The depth of the void areas corresponds to the total depth that you define in the machining cycle.

You can enter contours in the following ways:

Soft key	Function
CONTOUR <FILE> SELECT FILE	Define the name of the contour or Press the **SELECT FILE** soft key
CONTOUR <FILE>=QS	Define the number of a QS parameter
CONTOUR LBL NR	Define the number of a label
CONTOUR LBL NAME	Define the name of the label
CONTOUR LBL QS	Define the number of a QS parameter for a label

Example:

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

Programming notes:

- The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth Therefore, always start programming the subcontour with the deepest pocket.
- If the contour is defined as an island, the control interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
- If you enter a value of 0 for the depth, then the depth defined in Cycle **20** is 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. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Contour machining with SL Cycles

The entire contour is machined with the SL cycles (see "Overview", Page 282) or the OCM cycles (see "Overview", Page 337).

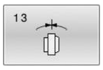
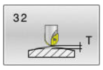
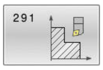
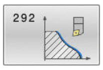
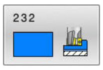
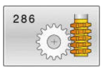
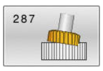
13

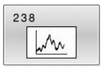
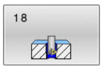
Cycles: Special Functions

13.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	Cycle 9 DWELL TIME ■ Delay execution by the programmed dwell time	444
	Cycle 12 PGM CALL ■ Call any NC program	445
	Cycle 13 ORIENTATION ■ Rotate spindle to a specific angle	447
	Cycle 32 TOLERANCE ■ Program the permissible contour deviation for jerk-free machining operations	448
	Cycle 291 COUPLG.TURNG.INTERP. (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Or, rescind the spindle coupling	452
	Cycle 292 CONTOUR.TURNG.INTRP. (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Create certain rotationally symmetric contours in the active working plane ■ Possible with tilted machining plane	459
	Cycle 225 ENGRAVING ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	469
	Cycle 232 FACE MILLING ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	476
	Cycle 285 DEFINE GEAR (option 157) ■ Define the geometry of the gear wheel	485
	Cycle 286 GEAR HOBBING (option 157) ■ Definition of the tool data ■ Selection of the machining strategy and side ■ Possibility of using the entire cutting edge	488
	Cycle 287 GEAR SKIVING (option 157) ■ Definition of the tool data ■ Selection of the machining side ■ Definition of the first and last infeed ■ Definition of the number of cuts	496

Soft key	Cycle	Page
	Cycle 238 MEASURE MACHINE STATUS (option 155) ■ Determine the current machine status or test the measuring sequence	508
	Cycle 239 ASCERTAIN THE LOAD (option 143) ■ Selection for a weighing run ■ Reset the load-dependent feedforward and controller parameters	511
	Cycle 18 THREAD CUTTING ■ With controlled spindle ■ Spindle stops at the bottom of the hole	514

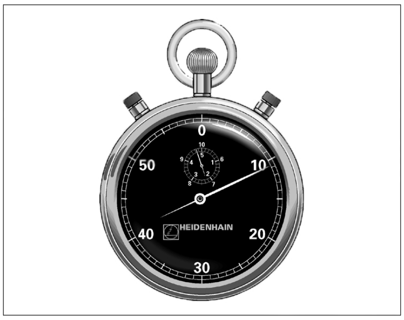
13.2 Cycle 9 DWELL TIME

ISO programming

G4

Application

> This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining mode.



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: User's Manual for **Klartext Programming**
- Dwell time with **FUNCTION DWELL**
  Further information: User's Manual for **Klartext Programming**

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

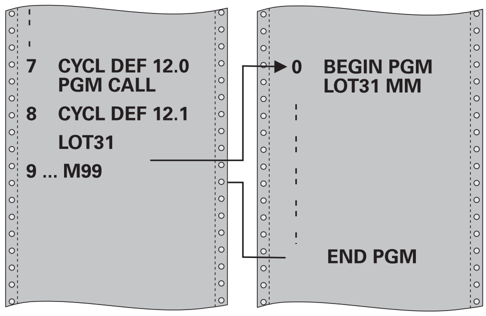
```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

13.3 Cycle 12 PGM CALL

ISO programming
G39

Application

NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.



Related topics

- Calling external NC programs
  Further information: User's Manual for **Klartext Programming**

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining mode.
- As a rule, Q parameters are globally effective when called with Cycle **12**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

Notes on programming

- The NC program you are calling must be stored in the internal memory of your control.
- If the NC program you are defining to be a cycle is located in the same directory as the NC program you are calling it from, you need only enter the program name.
- If the NC program you are defining to be a cycle is not located in the same directory as the NC program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.
- If you want to define an ISO program to be a cycle, add the .I file type to the program name.

Cycle parameters

Help graphic	Parameter
	Program name Enter the name of the NC program to be called and, if necessary, the path where it is located, Use the Select soft key to activate the File Select dialog. Select the NC program to be called. The **SYNTAX** soft key allows you to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path. If the complete path is within the quotation marks, you can use both **** and **/** to separate the folders and files.

Call the NC program with:

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

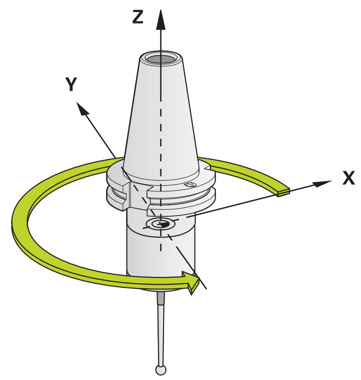
Declare NC program 1_Plate.h as a cycle and call it with M99

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

13.4 Cycle 13 ORIENTATION

ISO programming
G36

Application



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

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

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the 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

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining mode.
- 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**

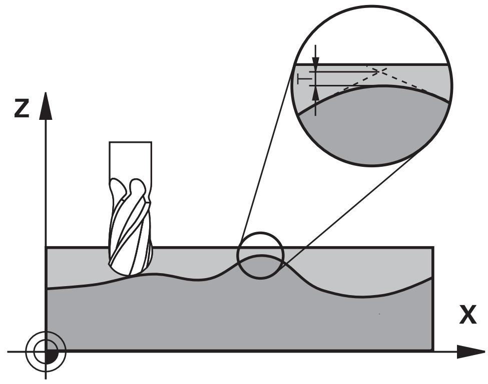
Example

```
11 CYCL DEF 13.0 ORIENTATION
12 CYCL DEF 13.1 ANGLE180
```

13.5 Cycle 32 TOLERANCE

ISO programming
G62

Application



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

With the entries in Cycle **32** you can influence the result of HSC machining with respect to accuracy, surface definition and speed, in as much as the control has been adapted to the machine's characteristics.

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

If necessary, the control automatically reduces the programmed feed rate so that the program can be executed at the fastest possible speed without jerking. **Even if the control does not move the axes with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the control can move the axes.

Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**tolerance value**) is set in a machine parameter by the machine manufacturer. With Cycle **32** you can change the pre-set tolerance value and select different filter settings, provided that your machine 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

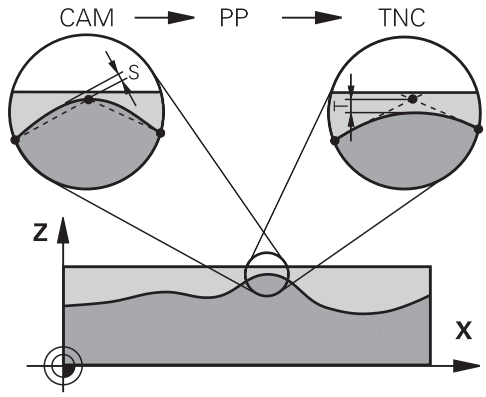
The control resets Cycle **32** if you do one of the following:

- 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: User's Manual for **Klartext Programming**

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining mode.
- 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 entered tolerance value **T** in millimeters. In an inch program it interprets it 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**

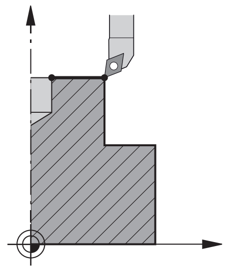
Example

```
11 CYCL DEF 32.0 TOLERANCE
12 CYCL DEF 32.1 T0.02
13 CYCL DEF 32.2 HSC-MODE:1 TA5
```

13.6 Cycle 291 COUPLG.TURNG.INTERP. (option 96)

ISO programming
G291

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

Cycle **291 COUPLG.TURNG.INTERP.** couples the tool spindle to the position of the linear axes, or cancels this spindle coupling. With interpolation turning, the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates **Q216** and **Q217**.

Cycle sequence

Q560=1:

1 The control first performs a spindle stop (**M5**).
2 The control orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation **Q336** is taken into account. If an "ORI" value is given in the tool table, it is also taken into account.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 To terminate the cycle, the coupling must be deactivated by the operator. (With Cycle **291** or end of program/internal stop.)

Q560=0:

1 The control deactivates the spindle coupling.
2 The tool spindle is no longer coupled to the position of the linear axes.
3 The control ends machining with Cycle **291** COUPLG.TURNG.INTERP.
4 If **Q560**=0, parameters **Q336**, **Q216**, **Q217** are not relevant

Notes

> This cycle is effective only for machines with servo-controlled spindle.
>
> Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine manufacturer for further information.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **291** is CALL-active.
- This cycle can also be used in a tilted working plane.
- Remember that the axis angle must be equal to the tilt angle before the cycle call! Only then can the axis be correctly coupled.
- If Cycle **8 MIRRORING** is active, the control does **not** execute the interpolation turning cycle.
- If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.

Notes on programming

- Programming of M3/M4 is not required. To describe the circular motions of the linear axes, you can, for example, use **CC** and **C** blocks.
- When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.
- Program outside contours with a radius greater than 0.
- Program inside contours with a radius greater than the tool radius.
- In order to attain high contouring speeds for your machine, define a large tolerance with Cycle **32** before calling the cycle. Program Cycle **32** with HSC filter=1.
- After defining Cycle **291** and **CYCL CALL**, program the operation you wish to perform. To describe the circular motions of the linear axes, you can use linear or polar coordinates, for example. An example is provided at the end of this section.

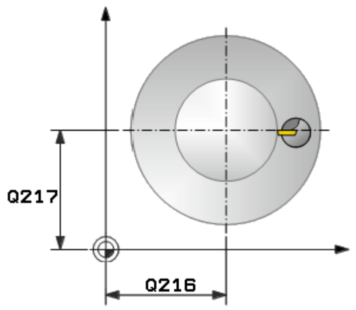
  Further information: "Example: Interpolation turning with Cycle 291", Page 516

Note regarding machine parameters

- In the machine parameter **mStrobeOrient** (no. 201005), the machine manufacturer defines the M function for spindle orientation.
  - If the value is > 0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine manufacturer). The control waits until the oriented spindle stop has been completed.
  - If you enter –1, the control will perform the oriented spindle stop.
  - If you enter 0, no action will be taken.

  The control will, under no circumstances, output **M5** before.

Cycle parameters

Help graphic	Parameter
	Q560 Spindle coupling (0=off, 1=on)? Define whether the tool spindle will be coupled to the position of the linear axes. If spindle coupling is active, the tool's cutting edge is oriented to the center of rotation. **0**: Spindle coupling off **1**: Spindle coupling on Input: **0**, **1**
	Q336 Angle for spindle orientation? The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that one cutting edge is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input: **0...360** **Further information:** "Defining the tool", Page 455
	**Q216 Center in 1st axis?** Center of rotation in the main axis of the working plane Absolute input: **-99999.9999...99999.9999**
	Q217 Center in 2nd axis? Center of rotation in the secondary axis of the working plane Input: **-99999.9999...+99999.9999**
	Q561 Convert turning tool (0/1) Only relevant if you define the turning tool in the turning tool table (toolturn.trn). This parameter allows you to decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool. **0**: No change; the turning tool is interpreted as described in the turning tool table (toolturn.trn). In this case, you must not use the radius compensation **RR** or **RL**. Furthermore, you must describe the movement of the path of the tool center point **TCP** without spindle coupling when programming. This kind of programming is much more complicated. **1:** The value XL from the turning tool table (toolturn.trn) is interpreted as a radius R of a milling tool table. This makes it possible to use radius compensation **RR** or **RL** when programming your contour. This kind of programming is recommended. Input: **0**, **1**

Example

```
11 CYCL DEF 291 COUPLG.TURNG.INTERP. ~
    Q560=+0                  ;SPINDLE COUPLING ~
    Q336=+0                  ;ANGLE OF SPINDLE ~
    Q216=+50                 ;CENTER IN 1ST AXIS ~
    Q217=+50                 ;CENTER IN 2ND AXIS ~
    Q561=+0                  ;CONVERT FROM TURNING TOOL
```

Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either activate (**Q560**=1) or deactivate (**Q560**=0) the COUPLG.TURNG.INTERP. cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

> **Q560**=0: Disable the **COUPLG.TURNG.INTERP.** cycle!

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter **Q560**=1, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling tool in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece during inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  - Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case, the orientation of the spindle to the center of rotation takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table), parameter **Q336**, and parameter **Q561**.

Programming and operating notes:

- If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter **Q561**=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With **Q561**=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program **Q561**=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

  If you programmed parameter **Q561**=1, you must program the following in order to conclude the interpolation turning machining operation:
  - R0, cancels radius compensation
  - Cycle **291** with parameters **Q560**=0 and **Q561**=0, deactivates spindle coupling
  - **CYCL CALL**, for calling Cycle **291**
  - **TOOL CALL** overrides the conversion of parameter **Q561**

  If you programmed parameter **Q561**=1, you may only use the following types of tools:
  - **TYPE**: **ROUGH**, **FINISH**, **BUTTON** with the machining directions **TO**: 1 or 8, **XL**>=0
  - **TYPE**: **ROUGH**, **FINISH**, **BUTTON** with the machining directions **TO**: 7: **XL**<=0

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	**ORI** + **Q336**
Interpolation turning, inside	7	**ORI** + **Q336** + 180
Interpolation turning, outside	7	**ORI** + **Q336** + 180
Interpolation turning, inside	1	**ORI** + **Q336**
Interpolation turning, outside	8	**ORI** + **Q336**
Interpolation turning, inside	8	**ORI** + **Q336**

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1, 7, 8
- TYPE: FINISH, with the machining directions TO: 1, 7, 8
- TYPE: BUTTON, with the machining directions TO: 1, 7, 8

The following tool types cannot be used for interpolation turning:

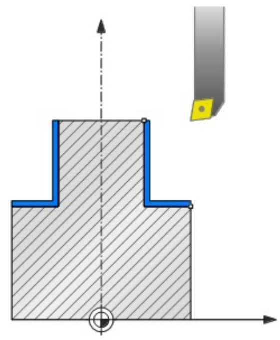
- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

13.7 Cycle 292 CONTOUR.TURNG.INTRP. (option 96)

ISO programming
G292

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

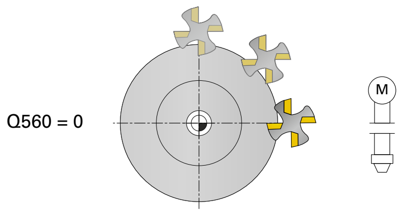


Cycle **292 INTERPOLATION TURNING CONTOUR FINISHING** couples the tool spindle to the positions of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. After executing this cycle, the control deactivates the spindle coupling again.

Before using Cycle **292**, you first need to define the desired contour in a subprogram and reference this contour with Cycle **14** or **SEL CONTOUR**. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. If you enter **Q560**=1, you can turn the contour and the cutting edge is oriented toward the circle center. If you enter **Q560**=0, you can mill the contour and the spindle is not oriented toward the circle center.

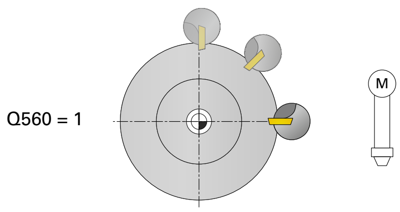
Cycle sequence

Cycle Q560=0: Contour milling



1 The M3/M4 function programmed before the cycle call remains in effect.
2 No spindle stop and **no** spindle orientation will be performed. **Q336** is not taken into account
3 The control positions the tool at the contour start radius **Q491**, taking the selected machining type (inside/outside, Q529) and the set-up clearance to the side (**Q357**) into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4 The control machines the defined contour using a rotating spindle (M3/M4). The principal axes of the working plane move along a circular path, whereas the spindle axis does not follow.
5 At the end point of the contour, the control retracts the tool perpendicularly to set-up clearance.
6 Finally, the control retracts the tool to the clearance height.

Cycle Q560=1: Contour turning



1 The control orients the tool spindle to the specified center of rotation. The specified angle **Q336** is taken into account. If an "ORI" value is given in the turning tool table (toolturn.trn), it is also taken into account.
2 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
3 The control positions the tool at the contour start radius **Q491**, taking the selected machining type (inside/outside, **Q529**) and the set-up clearance to the side (**Q357**) into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4 The control uses the interpolation turning cycle to machine the defined contour. In interpolation turning, the linear axes of the working plane move along a circular path, whereas the spindle axis follows, it is oriented perpendicularly to the surface.
5 At the end point of the contour, the control retracts the tool perpendicularly to set-up clearance.
6 Finally, the control retracts the tool to the clearance height.
7 The control automatically deactivates the coupling of the tool spindle to the linear axes.

Notes

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

Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine manufacturer for further information.

NOTICE

Danger of collision!

There is a risk of collision between tool and workpiece. The control does not automatically extend the described contour by a set-up clearance! At the beginning of the machining operation, the control positions the tool at rapid traverse FMAX to the contour starting point!

- Program an extension of the contour in the subprogram
- Make sure that there is no material at the contour starting point
- The center of the turning contour is the starting point in the working plane at the time the cycle is called

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The cycle is CALL-active.
- Roughing operations with multiple passes are not possible in this cycle.
- For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour **Q491** plus the set-up clearance to the side **Q357**. If the control determines that the tool is too large, the NC program will be canceled.
- Remember that the axis angle must be equal to the tilt angle before the cycle call! Only then can the axis be correctly coupled.
- If Cycle **8 MIRRORING** is active, the control does **not** execute the interpolation turning cycle.
- If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.
- In parameter **Q449 FEED RATE**, you program the feed rate at the starting radius. Keep in mind that the feed rate in the status display is referenced to the **TCP** and may deviate from **Q449**. The control calculates the feed rate in the status display as follows.

Outside machining **Q529 = 1**	Inside machining **Q529 = 0**
$F_{TCP} = Q449 \times \frac{(Q491 + R)}{Q491}$	$F_{TCP} = Q449 \times \frac{(Q491 - R)}{Q491}$

Notes on programming

- Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.
- Please note that it is not possible to define programmed finishing allowances via the **FUNCTION TURNDATA CORR-TCS(WPL)** function. Program a finishing allowance for your contour directly in the cycle or by specifying a tool compensation (DXL, DZL, DRS) in the tool table.
- When programming, remember to use only positive radius values.
- When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.
- Program outside contours with a radius greater than 0.
- Program inside contours with a radius greater than the tool radius.
- In order to attain high contouring speeds for your machine, define a large tolerance with Cycle **32** before calling the cycle. Program Cycle **32** with HSC filter=1.
- If you deactivate the spindle coupling (**Q560 = 0**), you can execute this cycle with polar kinematics. This requires that you clamp the workpiece at the center of the rotary table.

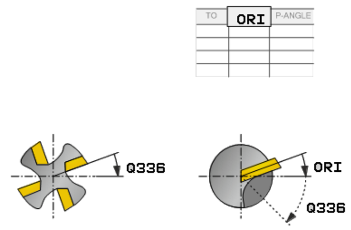
  Further information: User's Manual for **Klartext Programming**

Note regarding machine parameters

- With **Q560**=1, the control does not check whether the cycle is run with a rotating or stationary spindle. (Independent of **CfgGeoCycle** - **displaySpindleError** (no. 201002))
- In the machine parameter **mStrobeOrient** (no. 201005), the machine manufacturer defines the M function for spindle orientation.
  - If the value is > 0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine manufacturer). The control waits until the oriented spindle stop has been completed.
  - If you enter –1, the control will perform the oriented spindle stop.
  - If you enter 0, no action will be taken.

  The control will, under no circumstances, output **M5** before.

Cycle parameters

Help graphic	Parameter
	Q560 Spindle coupling (0=off, 1=on)? Define whether the spindle will be coupled or not. **0**: Spindle coupling off (mill the contour) **1**: Spindle coupling on (turn the contour) Input: **0...1**
	**Q336 Angle for spindle orientation?** The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that one cutting edge is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input: **0...360**
	Q546 Reverse tool rotation direction? Direction of spindle rotation of the active tool: **3**: Clockwise rotating tool (M3) **4**: Counter-clockwise rotating tool (M4) Input: **3**, **4**
	Q529 Machining operation (0/1)? Define whether an inside or outside contour will be machined: **+1**: Inside machining **0**: Outside machining Input: **0**, **1**
	Q221 Oversize for surface? Allowance in the working plane Input: **0...99.999**
	Q441 Infeed per revolution [mm/rev]? Dimension by which the control moves the tool during one revolution. Input: **0.001...99.999**
	Q449 Feed rate / cutting speed? (mm/min) Feed rate relative to the contour starting point **Q491**. The feed rate of the tool center point path is adjusted depending on the tool radius and **Q529 MACHINING OPERATION**. From these parameters, the control determines the programmed cutting speed at the diameter of the contour starting point. **Q529 = 1**: Feed rate of the tool center point path is reduced for inside machining. **Q529 = 0**: Feed rate of the tool center point path is increased for outside machining. Input: **1...99999** or **FAUTO**

Help graphic	Parameter
	Q491 Contour starting point (radius)? Radius of the contour starting point (e.g., X coordinate, if tool axis is Z). This value has an absolute effect. Input: **0.9999...99999.9999**
	Q357 Safety clearance to the side? Set-up clearance to the side of the workpiece when the tool approaches the first plunging depth. This value has an incremental effect. Input: **0...99999.9999**
	Q445 Clearance height? Absolute height at which collision between tool and workpiece is impossible. The tool retracts to this position at the end of the cycle. Input: **-99999.9999...+99999.9999**
	Q592 Type of dimension (0/1)? Interpretation of the contour dimensions: **0**: The control interprets the contour in the **ZX** coordinate plane. The control interprets the X axis values as radii. The coordinate system is left-handed. Therefore, the programmed direction of rotation for circles is as follows: ■ **DR-**: In clockwise direction ■ **DR+**: In counterclockwise direction **1**: The control interprets the contour in the **ZXØ** coordinate plane. The control interprets the X axis values as diameters. The coordinate system is right-handed. Therefore, the programmed direction of rotation for circles is as follows: ■ **DR-**: In counterclockwise direction ■ **DR+**: In clockwise direction Input: **0**, **1**

Example

```
11 CYCL DEF 292 CONTOUR.TURNG.INTRP. ~
    Q560=+0                ;SPINDLE COUPLING ~
    Q336=+0                ;ANGLE OF SPINDLE ~
    Q546=+3                ;CHANGE TOOL DIRECTN. ~
    Q529=+0                ;MACHINING OPERATION ~
    Q221=+0                ;SURFACE OVERSIZE ~
    Q441=+0.3              ;INFEED ~
    Q449=+2000             ;FEED RATE ~
    Q491=+50               ;CONTOUR START RADIUS ~
    Q357=+2                ;CLEARANCE TO SIDE ~
    Q445=+50               ;CLEARANCE HEIGHT ~
    Q592=+1                ;TYPE OF DIMENSION
```

Machining variants

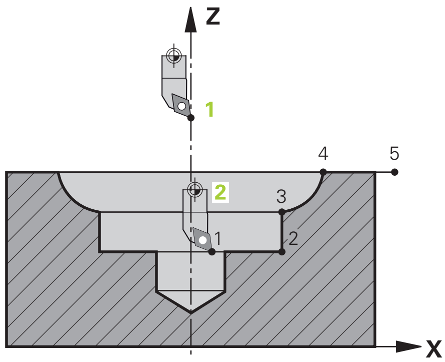
Before using Cycle **292**, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle **14** or **SEL CONTOUR**. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

Tool axis used	Axial coordinate	Radial coordinate
Z	Z	X
X	X	Y
Y	Y	Z

Example: If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius or diameter of the contour in X.

You can use this cycle for inside and outside machining. Some of the notes given in chapter "Notes", Page 461 are illustrated in the following. You will also find an example in "Example: Interpolation turning with Cycle 292", Page 519

Inside machining



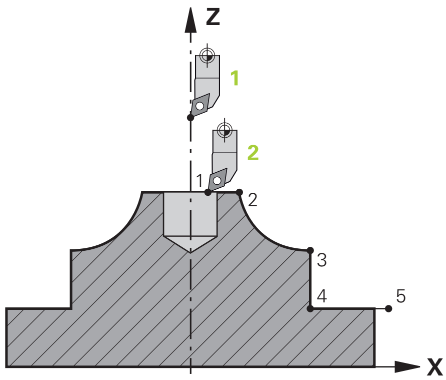
- The center of rotation is the position of the tool in the working plane when the cycle is called (**1**)
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! (**2**)
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control positions the tool to the contour starting point at rapid traverse in the tool axis direction. **Make sure that there is no material at the contour starting point.**

  You also need to take the following into account when programming the inside contour:
  - Program either monotonously increasing radial and axial coordinates (e.g., 1 to 5)
  - Or program monotonously decreasing radial and axial coordinates (e.g., 5 to 1)
  - Program inside contours with a radius greater than the tool radius.

Outside machining

- The center of rotation is the position of the tool in the working plane when the cycle is called (**1**)
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! (**2**)
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control positions the tool to the contour starting point at rapid traverse in the tool axis direction. **Make sure that there is no material at the contour starting point.**

  You also need to take the following into account when programming the outside contour:
  - Program either monotonously increasing radial coordinates and monotonously decreasing axial coordinates (e.g., 1 to 5)
  - Or program monotonously decreasing radial coordinates and monotonously increasing axial coordinates (e.g., 5 to 1)
  - Program outside contours with a radius greater than 0.



Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either mill (**Q560**=0) or turn (**Q560**=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling tool in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece during inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  ▶ Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**
  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.
- **Define a turning tool in the turning tool table (toolturn.trn)**
  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case, the orientation of the spindle to the center of rotation takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and parameter **Q336**.
  The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + **Q336**
Interpolation turning, inside	7	ORI + **Q336** + 180
Interpolation turning, outside	7	ORI + **Q336** + 180
Interpolation turning, inside	1	ORI + **Q336**
Interpolation turning, outside	8,9	ORI + **Q336**
Interpolation turning, inside	8,9	ORI + **Q336**

You can use the following tool types for interpolation turning:

- **TYPE**: **ROUGH**, with the machining directions **TO**: 1 or 7
- **TYPE**: **FINISH**, with the machining directions **TO**: 1 or 7
- **TYPE**: **BUTTON**, with the machining directions **TO**: 1 or 7

The following tool types cannot be used for interpolation turning:

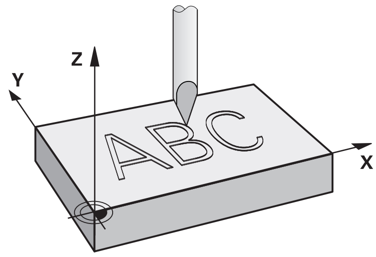
- **TYPE**: **ROUGH**, with the machining directions **TO**: 2 to 6
- **TYPE**: **FINISH**, with the machining directions **TO**: 2 to 6
- **TYPE**: **BUTTON**, with the machining directions **TO**: 2 to 6
- **TYPE**: **RECESS**
- **TYPE**: **RECTURN**
- **TYPE**: **THREAD**

13.8 Cycle 225 ENGRAVING

ISO programming
G225

Application

This cycle is used to engrave texts on a flat surface of the workpiece. You can arrange the texts in a straight line or along an arc.



Cycle sequence

1. If the tool is beneath **Q204 2ND SET-UP CLEARANCE**, the control will first move to the value from **Q204**.
2. The control positions the tool in the working plane to the starting point of the first character.
3. The control engraves the text.
   - If **Q202 MAX. PLUNGING DEPTH** is greater than **Q201 DEPTH**, the control will engrave each character in a single infeed motion.
   - If **Q202 MAX. PLUNGING DEPTH** is less than **Q201 DEPTH**, the control will engrave each character in several infeed motions. The control will always complete the milling of a character before machining the next one.
4. After the control has engraved a character, it retracts the tool to the set-up clearance **Q200** above the workpiece surface.
5. The process steps 2 and 3 are repeated for all characters to be engraved.
6. Finally, the control retracts the tool to 2nd set-up clearance **Q204**.

Notes

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

Notes on programming

- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The text to be engraved can also be transferred with a string variable (**QS**).
- Parameter **Q347** influences the rotational position of the letters.
  If **Q374** = 0° to 180°, the characters are engraved from left to right.
  If **Q374** is greater than 180°, the direction of engraving is reversed.

Cycle parameters

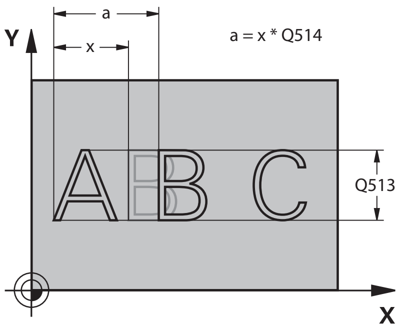
Help graphic | **Parameter**

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

Further information: "Engraving system variables", Page 474



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.

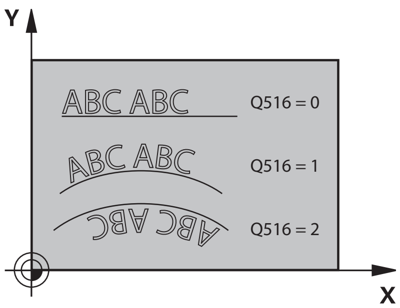
Input: **0...10**

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?

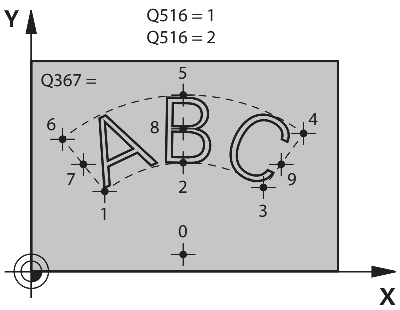
Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

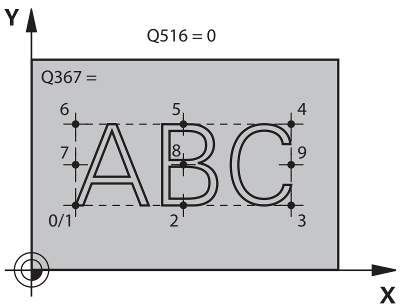
Input: **-99999.9999...+99999.9999**

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)?

Enter the reference for the position of the text here. Depending on whether the text will be engraved along a circular arc or in a straight line (parameter **Q516**), the following values can be entered:

Circle	Straight line
0 = Circle center	0 = Bottom left
1 = Bottom left	1 = Bottom left
2 = Bottom center	2 = Bottom center
3 = Bottom right	3 = Bottom right
4 = Top right	4 = Top right
5 = Top center	5 = Top center
6 = Top left	6 = Top left
7 = Center left	7 = Center left
8 = Center of text	8 = Center of text
9 = Center right	9 = Center right

Input: **0...9**

Help graphic	Parameter
	Q574 Maximum text length? Enter the maximum text length. The control also takes into account parameter **Q513** Character height. If **Q513 = 0**, the control engraves the text over exactly the length indicated in parameter **Q574**. The character height will be scaled accordingly. If **Q513 > 0**, the control checks whether the actual text length exceeds the maximum text length entered in **Q574**. If that is the case, the control displays an error message. Input: **0...999.999**
	Q202 Maximum plunging depth? Maximum infeed depth per cut. The machining operation is performed in several steps if this value is less than **Q201**. Input: **0...99999.9999**

Example

```
11 CYCL DEF 225 ENGRAVING ~
    QS500=""                ;ENGRAVING TEXT ~
    Q513=+10                ;CHARACTER HEIGHT ~
    Q514=+0                 ;SPACE FACTOR ~
    Q515=+0                 ;FONT ~
    Q516=+0                 ;TEXT ARRANGEMENT ~
    Q374=+0                 ;ANGLE OF ROTATION ~
    Q517=+50                ;CIRCLE RADIUS ~
    Q207=+500               ;FEED RATE MILLING ~
    Q201=-2                 ;DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q367=+0                 ;TEXT POSITION ~
    Q574=+0                 ;TEXT LENGTH ~
    Q202=+0                 ;MAX. PLUNGING DEPTH
```

Allowed engraving characters

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

> The control uses the special characters **%** and **** for special functions. If you want to engrave these characters, enter them twice in the text to be engraved (e.g., **%%**).

When engraving German umlauts, ß, ø, @, or the CE character, enter the character % before the character to be engraved:

Input	Character
%ae	ä
%oe	ö
%ue	ü
%AE	Ä
%OE	Ö
%UE	Ü
%ss	ß
%D	ø
%at	@
%CE	CE

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. Do do so, enter **%time<x>**. **<x>** defines the format (e.g., 08 for DD.MM.YYYY.) (Identical to the **SYSSTR ID10321** function).

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9 (e.g., **%time08**).

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

Cycle **225** allows you to engrave the current counter reading (provided in the MOD menu).

To do so, program Cycle **225** as usual and enter the text to be engraved, for example: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following: 000000003

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

Operating notes

- In Test Run operating mode, the control simulates only the counter reading that you have specified directly in the NC program. The counter reading from the MOD menu is not taken into account.
- In the SINGLE BLOCK and FULL SEQ. operating modes, the control will take the counter reading from the MOD menu into account.

13.9 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

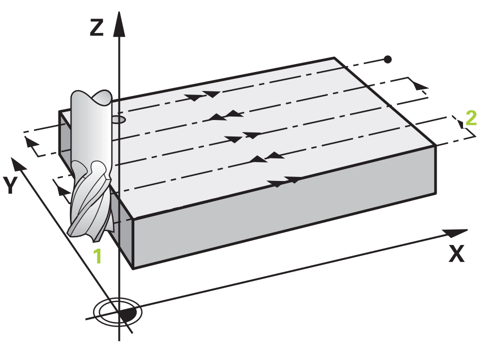
- Cycle **233 FACE MILLING**
  Further information: "Cycle 233 FACE MILLING ", Page 225

Cycle run

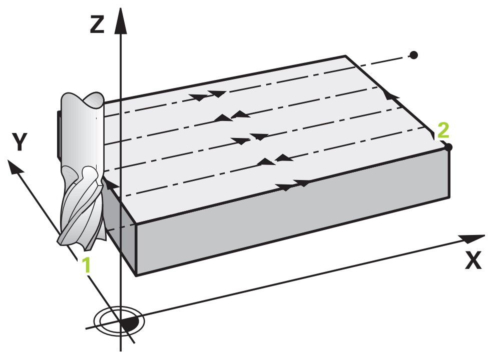
1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** using positioning logic: If the current position in the spindle axis is further away from the workpiece than the 2nd set-up clearance, the control positions the tool first in the working plane and then in the spindle axis. Otherwise, it first moves it to 2nd set-up clearance and then in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
2 The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0

3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

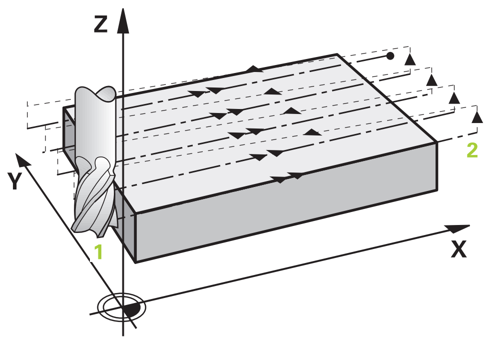


Strategy Q389=1



3 The tool subsequently advances at the programmed feed rate for milling to the end point 2. The end point lies **at the edge** of the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point 1. The motion to the next pass again occurs at the edge of the workpiece.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=2



3 The tool subsequently advances at the programmed feed rate for milling to the end point 2. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves it at the pre-positioning feed rate directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then returns to the current infeed depth and moves in the direction of end point 2
6 The process is repeated until the programmed surface has been machined completely. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Notes

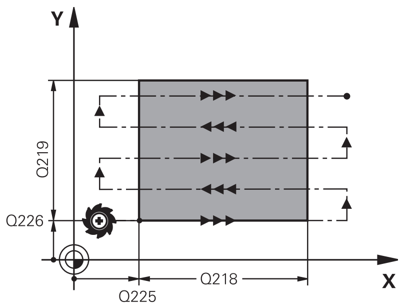
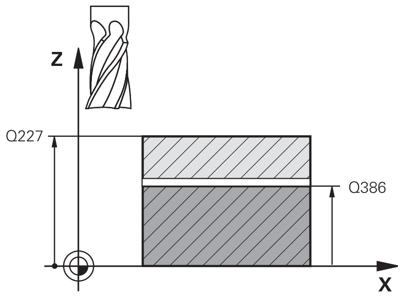
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

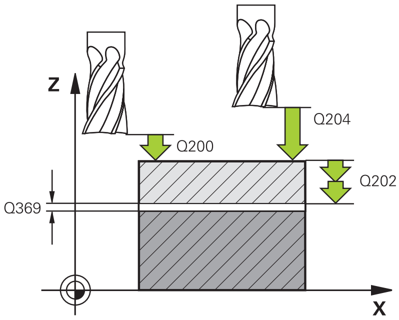
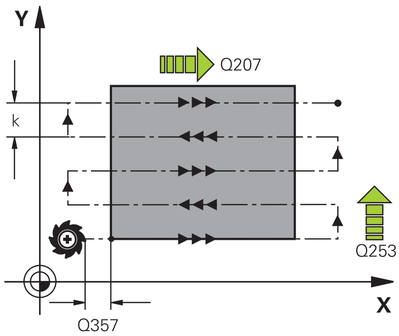
Notes on programming

- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- Program **Q227** greater than **Q386**. The control will otherwise display an error message.

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

Cycle parameters

Help graphic	Parameter
	Q389 Machining strategy (0/1/2)? Define how the control will machine the surface: **0**: Meander machining, stepover at positioning feed rate outside the surface to be machined **1**: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined **2**: Line-by-line machining, retraction and stepover at the positioning feed rate Input: **0**, **1**, **2**
 	**Q225 Starting point in 1st axis?** Define the starting point coordinate of the surface to be machined in the main axis of the working plane. 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**

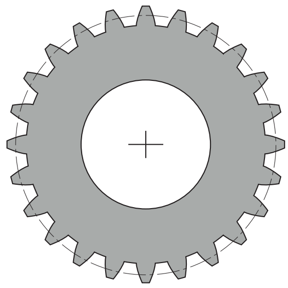
Help graphic	Parameter
	Q357 Safety clearance to the side? Parameter **Q357** influences the following situations: **Approaching the first infeed depth: Q357** is the lateral distance from the tool to the workpiece. **Roughing with the Q389 = 0 to 3 roughing strategies:** The surface to be machined is 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
```

13.10 Fundamentals for the machining of gear teeth (option 157)

Fundamentals



Refer to your machine manual.

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

The cycles require the gear cutting software option (option 157). When using these cycles in turning mode, the Mill-Turning software option (option 50) is additionally required. In milling mode, the tool spindle is the master spindle; in turning mode, it is the workpiece spindle. The other spindle is called slave spindle. Depending on the operating mode, you program the speed or the cutting speed with a **TOOL CALL S** or **FUNCTION TURNDATA SPIN**.

To orient the I-CS coordinate system, Cycles **286** and **287** use the precession angle that is also affected by Cycles **800** and **801** in turning mode. At the end of the cycle, the control resets the precession angle to its state at the beginning of the cycle. If one of these cycles is aborted, the precession angle will also be reset.

The axis crossing angle is the angle between workpiece and tool. It results from the angle of inclination of the tool and the angle of inclination of the gear. Based on the required axis crossing angle, Cycles **286** and **287** calculate the required inclination of the rotary axis at the machine. The cycles will always position the first rotary axis starting from the tool.

The cycles control **LIFTOFF** automatically to enable moving the tool out of the gear safely in case of fault. The cycles define the direction and the path for **LIFTOFF**. You only need to activate **LIFTOFF** for your tool. The machine manufacturer can configure the automatic **LIFTOFF**.

The gear itself will first be described in Cycle **285 DEFINE GEAR**. Then, program Cycle **286 GEAR HOBBING** or Cycle **287 GEAR SKIVING**.

Program the following:

- Call a tool with **TOOL CALL**
- Select turning mode or milling mode, with **FUNCTION MODE TURN** or **FUNCTION MODE MILL** "**KINEMATIC_GEAR**" kinematics selection
- Spindle direction of rotation (e.g., **M3** or **M303**)
- Perform pre-positioning for the cycle depending on your selection of **MILL** or **TURN**
- Define the **CYCL DEF 285 DEFINE GEAR** cycle
- Define the **CYCL DEF 286 GEAR HOBBING** or **CYCL DEF 287 GEAR SKIVING** cycle.

Notes

NOTICE
Danger of collision!
If you do not pre-position the tool to a safe position, a collision between tool and workpiece (fixtures) may occur during tilting.
▶ Pre-position the tool to a safe position

NOTICE
Danger of collision!
If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q200**!
▶ Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

- Before calling the cycle, set the preset to the center of rotation of the workpiece spindle.
- Please note that the slave spindle will continue to rotate after the end of the cycle. If you want to stop the spindle before the end of the program, make sure to program a corresponding M function.
- Activate the **LiftOff** in the tool table. In addition, this function must have been configured by your machine manufacturer.
- Remember that you need to program the speed of the master spindle before calling the cycle, i.e. the tool spindle speed in milling mode and the workpiece spindle speed in turning mode.

Gear formulas

Speed calculation

- n_T: Tool spindle speed
- n_W: Workpiece spindle speed
- z_T: Number of tool teeth
- z_W: Number of workpiece teeth

Definition	Tool spindle	Workpiece spindle
Hobbing	$n_T = n_W * z_w$	$n_W = \frac{n_T}{z_W}$
Skiving	$n_T = n_W * \frac{z_W}{z_T}$	$n_W = n_T * \frac{z_T}{z_W}$

Straight-cut spur gears

- m: Module (**Q540**)
- p: Pitch
- h: Tooth height (**Q563**)
- d: Pitch-circle diameter
- z: Number of teeth (**Q541**)
- c: Trough-to-tip clearance (**Q543**)
- d_a: Diameter of the addendum circle (outside diameter, **Q542**)
- d_f: Root circle diameter

Definition	Formula
Module (**Q540**)	$m = \frac{p}{\pi}$ $m = \frac{d}{z}$
Pitch	$p = \pi * m$
Pitch-circle diameter	$d = m * z$
Tooth height (**Q563**)	$h = 2 * m + c$
Diameter of the addendum circle (outside diameter, **Q542**)	$d_a = m * (z + 2)$ $d_a = d + 2 * m$
Root circle diameter	$d_f = d - 2 * (m + c)$
Root circle diameter if tooth height > 0	$d_f = d_a - 2 * (h + c)$
Number of teeth (**Q541**)	$z = \frac{d}{m}$ $z = \frac{d_a - 2 * m}{m}$

Remember to observe the algebraic sign when calculating an inner gear.

Example: Calculating the diameter of the addendum circle (outside diameter)

Outer gear: **Q540** * (**Q541** + 2) = 1 * (+46 + 2)

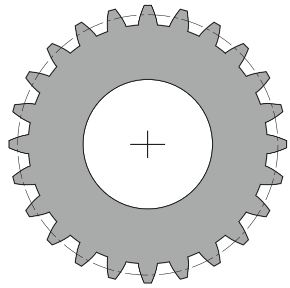
Inner gear: **Q540** * (**Q541** + 2) = 1 * (-46 + 2)

13.11 Cycle 285 DEFINE GEAR (option 157)

ISO programming
G285

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



Use Cycle **285 DEFINE GEAR** to describe the geometry of the gearing system. To describe the tool, use Cycle **286 GEAR HOBBING** or Cycle **287 GEAR SKIVING** and the tool table (TOOL.T).

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle is DEF-active. The values of these Q parameters will only be read when a CALL-active machining cycle is executed. If you overwrite these input parameters after the cycle definition and before calling the machining cycle, the gear geometry will be modified.
- Define the tool as a milling cutter in the tool table.

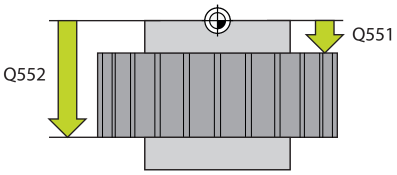
Notes on programming

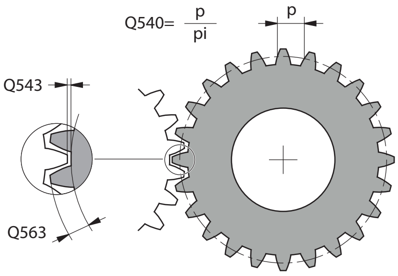
- You must specify values for module and number of teeth. If the outside diameter (diameter of the addendum circle) and the tooth height are defined as 0, normal running gears (DIN 3960) will be machined. If you want to machine gearing systems that differ from this standard, define the corresponding geometry by specifying the diameter of the addendum circle (outside diameter) **Q542** and the tooth height **Q563**.
- If the algebraic signs of the two input parameters **Q541** and **Q542** are contradictory, the cycle will be aborted with an error message.
- Remember that the diameter of the addendum circle is always greater than the root circle diameter, even for an inner gear.

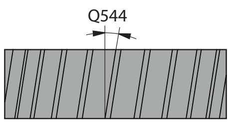
  Inner gear example: The outside diameter (addendum circle) is –40 mm, the root circle diameter is –45 mm. Also in this case, the diameter of the addendum circle (outside diameter) is (numerically) greater than the root circle diameter.

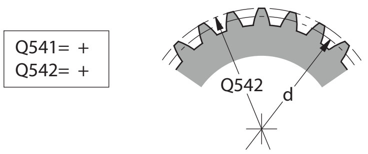
Cycle parameters

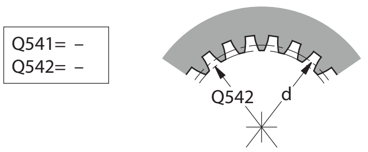
Help graphic











$Q541 = \frac{d}{Q540}$

Q542= Q540x(Q541+2)

Parameter

Q551 Starting point in Z?

Starting point of the hobbing process in Z

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

Q552 End point in Z?

End point of the hobbing process in Z

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

Q540 Module?

Module of the gear

Input: **0...99.999**

Q541 Number of teeth?

Number of teeth. This parameter depends on **Q542**.

+: If the number of teeth is positive, and at the same time the parameter **Q542** is positive, then an external gear will be machined.

-: If the number of teeth is negative, and at the same time the parameter **Q542** is negative, then an internal gear will be machined.

Input: **-99999...+99999**

Q542 Outside diameter?

Addendum circle (outside diameter) of the gear. This parameter depends on **Q541**.

+: If the addendum circle is positive, and at the same time the parameter **Q541** is positive, then an external gear will be machined.

–: If the addendum circle is negative, and at the same time the parameter **Q541** is negative, then an internal gear will be machined.

Input: **-9999.9999...+9999.9999**

Q563 Tooth height?

Distance from the tooth trough to the tooth tip.

Input: **0...999.999**

Q543 Trough-to-tip clearance?

Distance between the addendum circle of the gear to be made and root circle of the mating gear.

Input: **0...9.9999**

Q544 Angle of inclination?

Angle at which the teeth of a helical gear are inclined relative to the direction of the axis. For straight-cut gears, this angle is 0°.

Input: **-60...+60**

Example

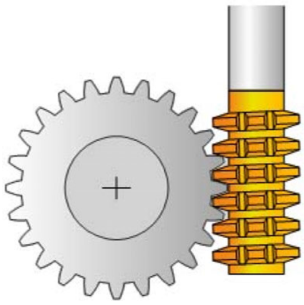
```
11 CYCL DEF 285 DEFINE GEAR ~
    Q551=+0              ;STARTING POINT IN Z ~
    Q552=-10             ;END POINT IN Z ~
    Q540=+1              ;MODULE ~
    Q541=+10             ;NUMBER OF TEETH ~
    Q542=+0              ;OUTSIDE DIAMETER ~
    Q563=+0              ;TOOTH HEIGHT ~
    Q543=+0.17           ;TROUGH-TIP CLEARANCE ~
    Q544=+0              ;ANGLE OF INCLINATION
```

13.12 Cycle 286 GEAR HOBBING (option 157)

ISO programming
G286

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



With Cycle **286 GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. You can select the machining strategy and the machining side in the cycle.
The machining process for gear hobbing is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction. Both for roughing and for finishing, the cutting operation may be offset by x edges relative to a height defined at the tool (e.g., 10 cutting edges for a height of 10 mm). This means that all cutting edges will be used in order to increase the tool life of the tool.

Related topics

- Cycle **880 GEAR HOBBING**
  Further information: "Cycle 880 GEAR HOBBING (option 50, option 131) ", Page 551

Cycle run

1. The control positions the tool in the tool axis to clearance height **Q260** at the feed rate **FMAX**. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2. Before tilting the working plane, the control positions the tool in X to a safe coordinate at the **FMAX** feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3. The control then tilts the working plane at the feed rate **Q253**
4. The control positions the tool at the feed rate **FMAX** to the starting point in the working plane
5. The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6. The control moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552**+**Q200** (**Q551** and **Q552** are defined in Cycle **285**).
   Further information: "Cycle 285 DEFINE GEAR (option 157)", Page 485
7. When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
8. The control repeats the steps 5 to 7 until the defined gear is completed.
9. Finally, the control retracts the tool to the clearance height **Q260** at the feed rate **FMAX**.

Notes

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

- Make sure to retract the tool before changing the position of the tilting axis

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- The maximum speed of the rotary table cannot be exceeded. If you have specified a higher value under **NMAX** in the tool table, the control will decrease the value to the maximum speed.

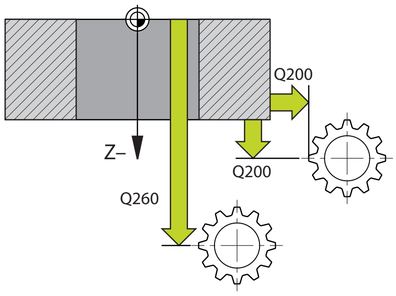
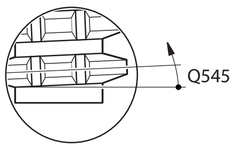
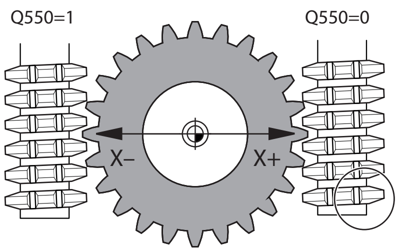
Avoid master spindle speeds of less than 6 rpm. Otherwise, it is not possible to reliably use a feed rate in mm/rev.

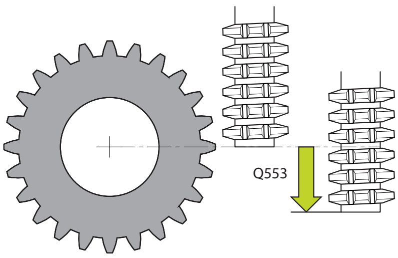
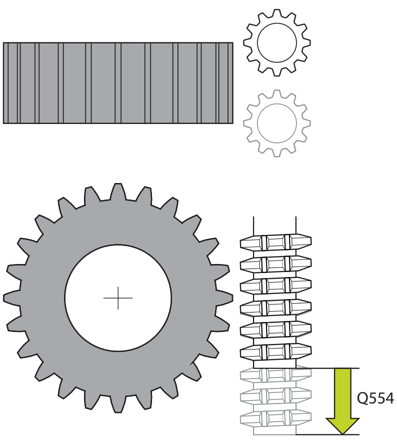
Notes on programming

- In order to ensure constant engagement of the cutting edge of a tool, you need to define a very small path in cycle parameter **Q554 SYNCHRONOUS SHIFT**.
- Make sure to program the direction of rotation of the master spindle (channel spindle) before the cycle start.
- If you program **FUNCTION TURNDATA SPIN VCONST:OFF S15**, the spindle speed of the tool is calculated as **Q541** x S. With **Q541** = 238 and S = 15, this would result in a tool spindle speed of 3570 rpm.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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**
	**Q545 Tool lead angle?** Angle of the edges of the gear hob. Enter this value in decimal notation. Example: 0°47'=0.7833 Input: **-60...+60**
	**Q546 Reverse spindle rotation dir.?** Direction of rotation of the slave spindle: **0**: No change in the direction of rotation **1**: Change in the direction of rotation Input: **0**, **1** **Further information:** "Verifying and changing directions of rotation of the spindles", Page 494
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**
	Q550 Machining side (0=pos./1=neg.)? Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**

Help graphic	Parameter
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	**Q253 Feed rate for pre-positioning?** Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q553 TOOL:L offset, machining start? Define the minimum length offset (L OFFSET) that the tool should have when in use. The control offsets the tool in the longitudinal direction by this amount. This value has an incremental effect. Input: **0...999.999**
	**Q554 Path for synchronous shift?** Define by which distance the gear hob will be offset in its axial direction during machining. This way, tool wear can be distributed over this area of the cutting edges. For helical gears, it is thus possible to limit the cutting edges used for machining. Entering **0** deactivates the synchronous shift function. Input: **-99...+99.9999**

Help graphic	Parameter
	Q548 Tool shift for roughing? Specify the number of cutting edges by which the control will shift the roughing tool in its axial direction. The shift will be performed incrementally relative to parameter **Q553**. Entering 0 deactivates the shift function. Input: **-99...+99**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0.001...999.999**
	Q488 Feed rate for plunging Feed rate for tool infeed. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q478 Roughing feed rate? Feed rate during roughing. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q549 Tool shift for finishing? Specify the number of cutting edges by which the control will shift the finishing tool in its longitudinal direction. The shift will be performed incrementally relative to parameter **Q553**. Entering 0 deactivates the shift function. Input: **-99...+99**

Example

```
11 CYCL DEF 286 GEAR HOBBING ~
    Q215=+0                ;MACHINING OPERATION ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q260=+100              ;CLEARANCE HEIGHT ~
    Q545=+0                ;TOOL LEAD ANGLE ~
    Q546=+0                ;CHANGE ROTATION DIR. ~
    Q547=+0                ;ANG. OFFSET, SPINDLE ~
    Q550=+1                ;MACHINING SIDE ~
    Q533=+0                ;PREFERRED DIRECTION ~
    Q530=+2                ;INCLINED MACHINING ~
    Q253=+750              ;F PRE-POSITIONING ~
    Q553=+10               ;TOOL LENGTH OFFSET ~
    Q554=+0                ;SYNCHRONOUS SHIFT ~
    Q548=+0                ;ROUGHING SHIFT ~
    Q463=+1                ;MAX. CUTTING DEPTH ~
    Q488=+0.3              ;PLUNGING FEED RATE ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q549=+0                ;FINISHING SHIFT
```

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Determine the direction of rotation of the rotary table:

1. What tool? (Right-cutting/left-cutting?)
2. Which machining side? **X+ (Q550=0) / X- (Q550=1)**
3. Look up the direction of rotation of the rotary table in one of the two tables below! To do so, select the appropriate table for the direction of rotation of your tool (right-cutting/left-cutting). Please refer to the appropriate table below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Clockwise (e.g., **M303**)
X- (Q550=1)	Counterclockwise (e.g., **M304**)

Tool: Left-cutting M4

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Counterclockwise (e.g., **M304**)
X- (Q550=1)	Clockwise (e.g., **M303**)

Keep in mind that in special cases, the directions of rotation might deviate from the ones indicated in these tables.

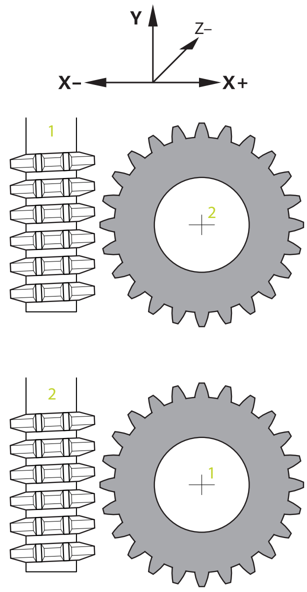
Changing the direction of rotation

Milling:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Turning:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.



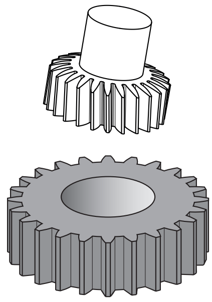
> Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.
>
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

13.13 Cycle 287 GEAR SKIVING (option 157)

ISO programming
G287

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



With Cycle **287 GEAR SKIVING**, you can machine cylindrical gears or helical gears with any angles. Cutting takes place on the one hand by the axial feeding of the tool and on the other hand through the rolling motion.

You can select the machining side in the cycle. The machining process for gear skiving is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.

In the cycle, you can call a table containing technology data. In this table, you can define a feed rate, a lateral infeed and a lateral offset or a specific tooth flank profile for each single cut.

Further information: "Table containing technology data", Page 503

Cycle run

1 The control positions the tool in the tool axis to the clearance height **Q260** at the feed rate **FMAX**. The tool will move only when the current position in the tool axis is below **Q260**.
2 Before tilting the working plane, the control positions the tool in X at the feed rate **FMAX** to a safe coordinate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control tilts the working plane at the feed rate **Q253**.
4 The control positions the tool to the starting point in the working plane at the feed rate **FMAX**.
5 Then the control moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6 The control approaches the approach length. The control automatically calculates this distance. The approach length is the distance from the initial scratch to the complete plunging depth.
7 The control rolls the tool over the workpiece to be geared in longitudinal direction at the defined feed rate. In the first infeed **Q586**, the control moves with the first feed rate **Q588**.
8 At the end of the cut, the tool moves beyond the defined end point by the overrun path **Q580**. The overrun path serves to completely machine the gear.
9 For further cuts, the control calculates the feed rate and the infeed itself.
  The calculated feed rate values depend on the feed rate adaptation factor **Q580**.
  The calculated infeed values are intermediate values of parameters **Q586 FIRST INFEED** and **Q587 LAST INFEED**.
10 The control executes the last infeed **Q587** at feed rate **Q589**.
11 When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
12 Finally, the control retracts the tool to the clearance height **Q260** at the feed rate **FMAX**.

- The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552** (**Q551** and **Q552** are defined in Cycle **285**). The approach length must be added to the starting point. Its purpose is to prevent the tool from plunging into the workpiece all the way to the machining diameter. The control calculates this distance itself.
- After every cut, the control displays a pop-up window showing the number of the current cut and the number of remaining cuts.

Notes

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

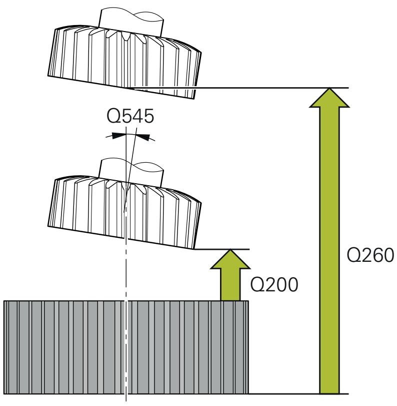
- Make sure to retract the tool before changing the position of the tilting axis

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- The speed ratio between tool and workpiece results from the number of teeth of the gear wheel and the number of cutting edges of the tool.

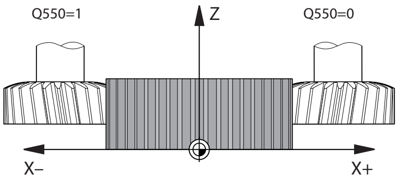
Notes on programming

- Make sure to program the direction of rotation of the master spindle (channel spindle) before the cycle start.
- The larger the factor in **Q580 FEED-RATE ADAPTION**, the earlier the control will adapt the feed rate to the feed rate for the last cut. The recommended value is 0.2.
- When defining the tool, make sure to specify the number of cutting edges as indicated in the tool table.
- If only two cuts have been programmed in **Q240**, the last infeed from **Q587** and the last feed rate from **Q589** will be ignored. If only one cut has been programmed, the first infeed from **Q586** will also be ignored.
- If the optional parameter **Q466 OVERRUN PATH** is programmed, the control optimizes the approach lengths and overrun path automatically to match the current cutting depth.

Cycle parameters

Help graphic	Parameter
	Q240 Number of cuts? Number of cuts to the final depth **0**: The control automatically determines the minimum number of cuts **1**: One cut **2**: Two cuts where the control considers only the infeed for the first cut **Q586**. The control does not consider the infeed for the last cut **Q587**. **3 to 99**: Programmed number of cuts **"..."**: Path of a table containing technology data see "Table containing technology data", Page 503 Input: **0...99** or text entry of max. **255** characters or **QS** parameter
	Q584 Number of the first cut? Define which cut number the control will perform first. Input: **1...999**
	Q585 Number of the last cut? Define at which number the control will perform the last cut. Input: **1...999**
	**Q200 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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**
	Q545 Tool lead angle? Angle of the edges of the skiving tool. Enter this value in decimal notation. Example: 0°47' = 0.7833 Input: **-60...+60**
	Q546 Reverse spindle rotation dir.? Direction of rotation of the slave spindle: **0**: No change in the direction of rotation **1**: Change in the direction of rotation Input: **0**, **1** **Further information:** "Verifying and changing directions of rotation of the spindles", Page 506
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**

Help graphic	Parameter



Q550 Machining side (0=pos./1=neg.)?

Define at which side machining is to take place.

0: Positive machining side of the main axis in the I-CS

1: Negative machining side of the main axis in the I-CS

Input: **0**, **1**

Q533 Preferred dir. of incid. angle?

Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use:

0: Solution that is the shortest distance from the current position

-1: Solution that is in the range between 0° and -179.9999°

+1: Solution that is in the range between 0° and +180°

-2: Solution that is in the range between -90° and -179.9999°

+2: Solution that is between +90° and +180°

Input: **-2**, **-1**, **0**, **+1**, **+2**

Q530 Inclined machining?

Position the tilting axes for inclined machining:

1: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes

2: Automatically position the tilting axis without orienting the tool tip (**TURN**)

Input: **1**, **2**

Q253 Feed rate for pre-positioning?

Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min.

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

Q586 Infeed for first cut?

Infeed for the first cut. This value has an incremental effect.

If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Table containing technology data", Page 503

Input: **0.001...99.999**

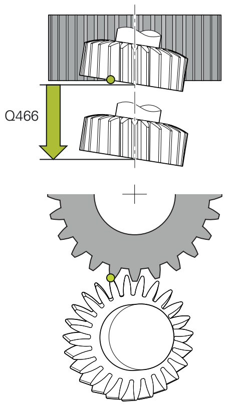
Q587 Infeed for last cut?

Infeed for the last cut. This value has an incremental effect.

If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Table containing technology data", Page 503

Input: **0.001...99.999**

Help graphic	Parameter
	Q588 Feed rate for first cut? Feed rate for the first cut. The control interprets the feed rate in mm per workpiece revolution. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Table containing technology data", Page 503 Input: **0.001...99.999**
	Q589 Feed rate for last cut? Feed rate for the last cut. The control interprets the feed rate in mm per workpiece revolution. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Table containing technology data", Page 503 Input: **0.001...99.999**
	Q580 Factor for feed-rate adaptation? Using this factor, you can define a feed rate reduction. This is due to the fact that the feed rate must decrease with increasing cutting numbers. The greater the value, the earlier the control will adapt the feed rates to match the last feed rate. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Table containing technology data", Page 503 Input: **0...1**

Help graphic	Parameter
	**Q466 Overrun path?** Length of overrun at the end of the gear The overtravel path ensures that the control machines the gear teeth up to the desired end point. The control automatically optimizes the overrun path to match the current cutting depth. When deleting this optional parameter with **NO ENT**, the control uses the set-up clearance **Q200** as the overrun path. In this case the control will not automatically optimize the overrun path. Input: **0.1...99.9**

Example

```
11 CYCL DEF 287 GEAR SKIVING ~
    Q240=+0             ;NUMBER OF CUTS ~
    Q584=+1             ;NO. OF FIRST CUT ~
    Q585=+999           ;NO. OF LAST CUT ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q260=+100           ;CLEARANCE HEIGHT ~
    Q545=+0             ;TOOL LEAD ANGLE ~
    Q546=+0             ;CHANGE ROTATION DIR. ~
    Q547=+0             ;ANG. OFFSET, SPINDLE ~
    Q550=+1             ;MACHINING SIDE ~
    Q533=+0             ;PREFERRED DIRECTION ~
    Q530=+2             ;INCLINED MACHINING ~
    Q253=+750           ;F PRE-POSITIONING ~
    Q586=+1             ;FIRST INFEED ~
    Q587=+0.1           ;LAST INFEED ~
    Q588=+0.2           ;FIRST FEED RATE ~
    Q589=+0.05          ;LAST FEED RATE ~
    Q580=+0.2           ;FEED-RATE ADAPTION ~
    Q466=+2             ;OVERRUN PATH
```

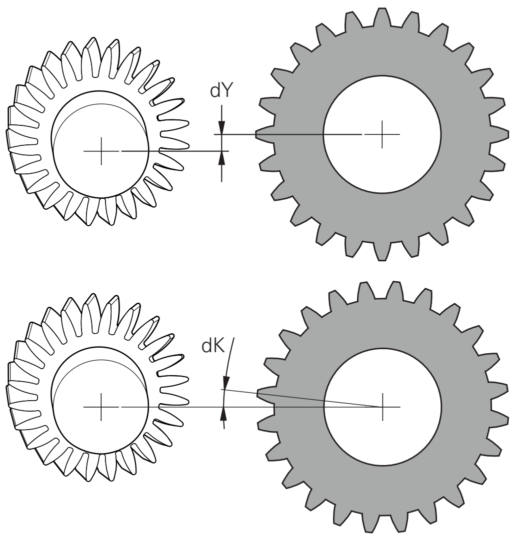
Table containing technology data

In Cycle **287 GEAR SKIVING**, you can use cycle parameter **QS240 NUMBER OF CUTS** to call a table containing technology data. The table is a freely definable table, and is in the ***.tab** format. The control provides you with a **Proto_Skiving.TAB** template. In the table, you define the following data for each individual cut:

- Feed rate
- Lateral infeed
- Lateral offset
- Angular offset of the workpiece
- If necessary, a profile program for an individual tooth flank line

Parameters in the table

The technology data table contains the following parameters:



Parameter	Function
NR	Number of the cut that also corresponds to the number of the table row
FEED	Feed rate in mm/rev or 1/10 inch/rev for the cut This parameter replaces the following cycle parameters: ■ **Q588 FIRST FEED RATE** ■ **Q589 LAST FEED RATE** ■ **Q580 FEED-RATE ADAPTION** Input: **0...9999.999**
INFEED	Lateral infeed of the cut. This entry is incremental. This parameter replaces the following cycle parameters: ■ **Q586 FIRST INFEED** ■ **Q587 LAST INFEED** Input: **0...99.99999**
dY	Lateral offset between tool and workpiece Use the **dY** offset to machine only one side of the tooth flank. In this way, it might be possible to increase the surface quality with **dY**. The entered values can lead to a distortion of the tooth flank profile, which might need to be considered in the profile of the cutting edges. Input: **-9.99999...+9.99999**
dK	Angular offset of the workpiece Use the **dK** angular offset to machine only one side of the tooth flank. It then might be possible to increase the surface quality. The entered values can lead to a distortion of the tooth flank profile, which might need to be considered in the profile of the cutting edges. Input: **-9.99999...+9.99999**

Parameter	Function
PGM	Profile program for an individual tooth flank line **Further information:** "Profile program of the tooth flank line", Page 505

Notes

- The unit used in the NC program determines whether millimeter or inch units are used.
- HEIDENHAIN recommends that you program only minimum offset values **dY** and minimum offsets **dK** in the individual cuts, in order to avoid damage to the contour.
- The two values **dY** and **dK** can be combined with each other.
- The sum of the lateral infeeds (**INFEED**) must result in the tooth height.
  - If the tooth height is greater than the total infeed, the control will display a warning.
  - If the tooth height is less than the total infeed, the control will display an error message.

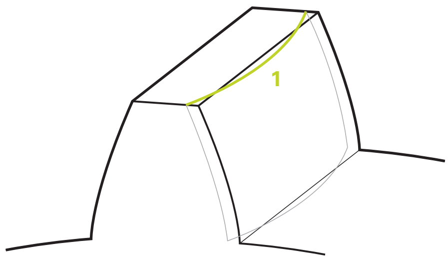
  Example:
  - **TOOTH HEIGHT** (**Q563**) = 2 mm
  - Number of cuts (**NR**) = 15
  - Lateral infeed (**INFEED**) = 0.2 mm
  - Total infeed = **NR** * **INFEED** = 3 mm

    In this case, the tooth height is less than the total infeed (2 mm < 3 mm).

    Reduce the number of cuts to 10.

Profile program of the tooth flank line

With a separate NC program you can define an individual tooth flank line **1**, such as a minimum crowning of the tooth flank.



You must pay attention to the following in the profile program:

- Do not program a feed rate.
- The cycle automatically calculates and executes pre-positioning and the overrun path.
- In turning mode, take an active diameter or radius programming into account.
- The datum for the profile program is at the starting point of the tooth flank.

> Use the **Q584 NO. OF FIRST CUT** parameter to read and evaluate the active cut number in the NC program.

Example application:

The finished gear wheels often transmit large forces when the teeth press against each other. These large forces can cause deformation of the material, for example, and thus lead to uneven load distribution on the tooth flank. The uneven load distribution can cause wear on the gear wheel. To reduce or avoid wear on the gear wheel, you can optimize the tooth flank line; for example, by adding minimum crowning on the tooth flank.

Further information: "Example of skiving with technology table and profile program", Page 525

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Determine the direction of rotation of the rotary table:

1 What tool? (Right-cutting/left-cutting?)
2 Which machining side? **X+ (Q550=0) / X- (Q550=1)**
3 Look up the direction of rotation of the rotary table in one of the two tables below! To do so, select the appropriate table for the direction of rotation of your tool (right-cutting/left-cutting). Please refer to the appropriate table below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Clockwise (e.g., **M303**)
X- (Q550=1)	Counterclockwise (e.g., **M304**)

Tool: Left-cutting M4

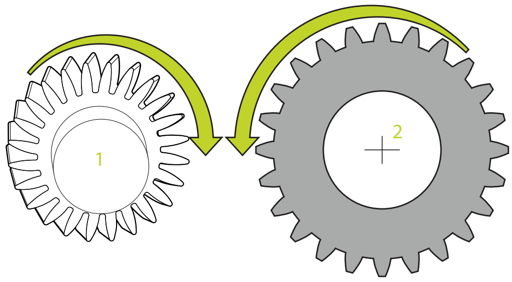
Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Counterclockwise (e.g., **M304**)
X- (Q550=1)	Clockwise (e.g., **M303**)

Keep in mind that in special cases, the directions of rotation might deviate from the ones indicated in these tables.

Changing the direction of rotation

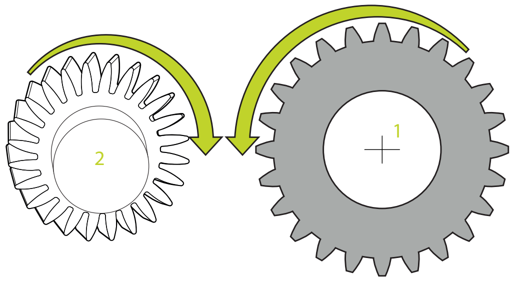
Milling:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.



Turning:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.



> Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.
>
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

13.14 Cycle 238 MEASURE MACHINE STATUS (option 155)

ISO programming
G238

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

During their lifecycle, the machine components which are subject to loads (e.g., guides, ball screws, ...) become worn and thus, the quality of the axis movements deteriorates. This, in turn, affects the production quality.

With software option **Component Monitoring** (option 155) and Cycle **238**, the control is able to measure the current machine status. As a result, any deviations from the machine's shipping condition due to wear and aging can be measured. The measurement results are stored in a text file that is readable for the machine manufacturer. He 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** (option 155)
  Further information: User's Manual for **Klartext Programming**

Cycle run

> 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 Detail** status tab, you can select the monitoring task to be displayed
4 This diagram allows you to watch how close the components are to a warning or error threshold

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

> Your machine 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**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** 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 and then press the **NC Start** key to run the cycle again.

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 potentiome-ters Input: **0**, **1**

Example

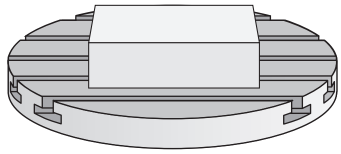
```
11 CYCL DEF 238  MEASURE MACHINE STATUS ~
    Q570=+0                    ;MODE
```

13.15 Cycle 239 ASCERTAIN THE LOAD (option 143)

ISO programming

G239

Application



> Refer to your machine manual.
>
> This function must be enabled and adapted by the machine manufacturer.

The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of the table axes. With software option **Load Adaptive Control** (option 143) and Cycle **239 ASCERTAIN THE LOAD**, the control is able to ascertain and adapt the current mass inertia of the load, the current friction forces and the maximum axis acceleration automatically or to reset 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 run

Parameter Q570 = 0

1 There is no physical movement of the axes.
2 The control resets the LAC.
3 The control activates feedforward and, if applicable, controller parameters that allow safe movements of the axis/axes, independently of the current load condition. The parameters set with **Q570**=0 are **independent** of the current load
4 These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

1 The control performs a weighing procedure in which it moves one or more axes. Which axes are moved depends on the setup of the machine and on the drives of the axes.
2 The scope of axis movement is defined by the machine manufacturer.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the ascertained parameters.

> If you are using the mid-program startup function and the control thus skips Cycle **239** in the block scan, the control will ignore this cycle—no weighing run will be performed.

Notes

NOTICE

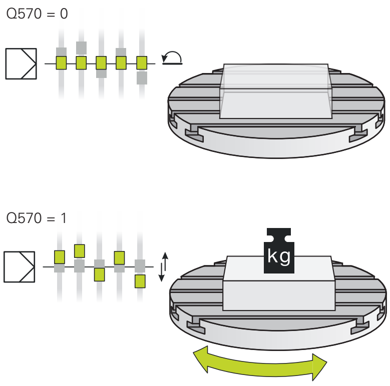
Danger of collision!

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**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** 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

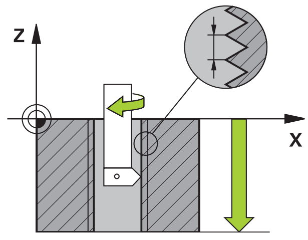
```
11 CYCL DEF 239  ASCERTAIN THE LOAD ~
    Q570=+0                 ;LOAD ASCERTATION
```

13.16 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: "Cycles: Tapping / Thread Milling", Page 129

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

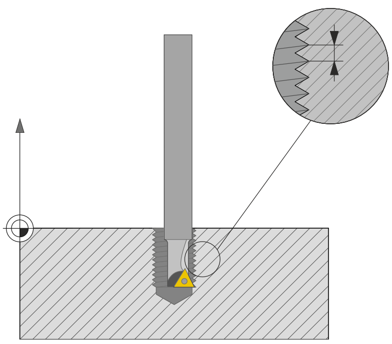
Notes on programming

- Before calling this cycle, program a spindle stop (for example with M5). The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.

Note regarding machine parameters

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

Cycle parameters

Help graphic	Parameter
	**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**

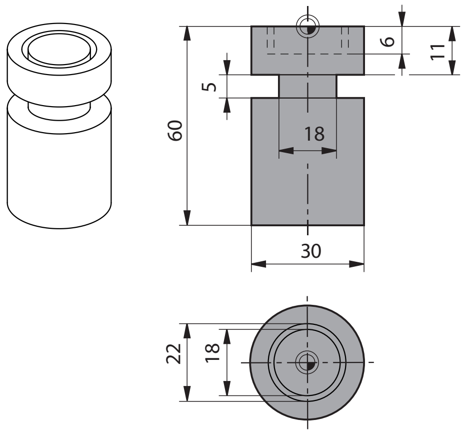
Example

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

13.17 Programming examples

Example: Interpolation turning with Cycle 291

The following NC program illustrates the use of Cycle **291 COUPLG.TURNG.INTERP.** This programming example shows how to machine an axial recess and a radial recess.



Tools

- Turning tool as defined in toolturn.trn: Tool no. 10: TO:1, ORI:0, TYPE:ROUGH; tool for axial recesses
- Turning tool as defined in toolturn.trn: Tool no. 11: TO:8, ORI:0, TYPE:ROUGH; tool for radial recesses

Program sequence

- Tool call: Tool for axial recess
- Start of interpolation turning: Description and call of Cycle **291**; **Q560** = 1
- End of interpolation turning: Description and call of Cycle **291**; **Q560** = 0
- Tool call: Recessing tool for radial recess
- Start of interpolation turning: Description and call of Cycle **291**; **Q560** = 1
- End of interpolation turning: Description and call of Cycle **291**; **Q560** = 0

> By converting parameter **Q561**, the turning tool is displayed in the simulation graphic as a milling tool.

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R15 L60	
2 TOOL CALL 10	; Tool call: tool for axial recess
3 CC X+0 Y+0	
4 LP PR+30 PA+0 R0 FMAX	; Retract the tool
5 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+1 ;CONVERT FROM TURNING TOOL	
6 CYCL CALL	; Call the cycle
7 LP PR+9 PA+0 RR FMAX	; Position the tool in the working plane
8 L Z+10 FMAX	
9 L Z+0.2 F2000	; Position the tool in the spindle axis
10 LBL 1	; Recessing on face (infeed: 0.2 mm, depth: 6 mm)
11 CP IPA+360 IZ-0.2 DR+ F10000	
12 CALL LBL 1 REP30	
13 LBL 2	; Retract from recess (step: 0.4 mm)
14 CP IPA+360 IZ+0.4 DR+	
15 CALL LBL 2 REP15	

16 L Z+200 R0 FMAX	; Retract to clearance height, deactivate radius compensation
17 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+0 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+0 ;CONVERT FROM TURNING TOOL	
18 CYCL CALL	; Call the cycle
19 TOOL CALL 11	; Tool call: tool for radial recess
20 CC X+0 Y+0	
21 LP PR+25 PA+0 R0 FMAX	; Retract the tool
22 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+1 ;CONVERT FROM TURNING TOOL	
23 CYCL CALL	; Call the cycle
24 LP PR+15 PA+0 RR FMAX	; Position the tool in the working plane
25 L Z+10 FMAX	
26 L Z-11 F7000	; Position the tool in the spindle axis
27 LBL 3	; Recessing on lateral surface (infeed: 0.2 mm, depth: 6 mm)
28 CC X+0.1 Y+0	
29 CP IPA+180 DR+ F10000	
30 CC X-0.1 Y+0	
31 CP IPA+180 DR+	
32 CALL LBL 3 REP15	
33 LBL 4	; Retract from recess (step: 0.4 mm)
34 CC X-0.2 Y+0	
35 CP PA+180 DR+	
36 CC X+0.2 Y+0	
37 CP IPA+180 DR+	
38 CALL LBL 4 REP8	
39 LP PR+50 FMAX	
40 L Z+200 R0 FMAX	; Retract to clearance height, deactivate radius compensation
41 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+0 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+0 ;CONVERT FROM TURNING TOOL	
42 CYCL CALL	; Call the cycle

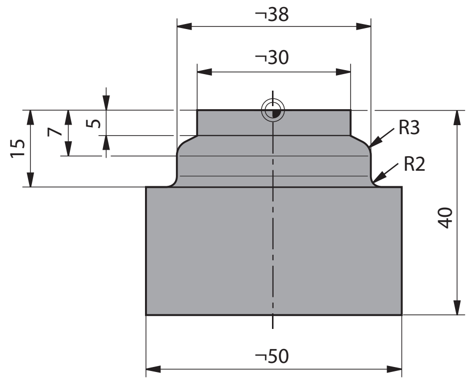
43 TOOL CALL 11	; Repeated **TOOL CALL** in order to reset the conversion of parameter Q561
44 M30	
45 END PGM 5 MM	

Example: Interpolation turning with Cycle 292

The following NC program illustrates the use of Cycle **292 CONTOUR.TURNG.INTRP.** This programming example shows how to machine an outside contour with the milling spindle rotating.

Program sequence

- Tool call: Milling cutter D20
- Cycle **32 TOLERANCE**
- Reference to the contour with Cycle **14**
- Cycle **292 CONTOUR.TURNG.INTRP.**



0 BEGIN PGM 6 MM	
1 BLK FORM CYLINDER Z R25 L40	
2 TOOL CALL 10 Z S111	; Tool call: end mill D20
*** - ...**	; Use Cycle 32 to define the tolerance
3 CYCL DEF 32.0 TOLERANZ	
4 CYCL DEF 32.1 T0.05	
5 CYCL DEF 32.2 HSC-MODE:1	
6 CYCL DEF 14.0 CONTOUR	
7 CYCL DEF 14.1 CONTOUR LABEL1	
8 CYCL DEF 292 CONTOUR.TURNG.INTRP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q546=+3 ;CHANGE TOOL DIRECTN. ~	
Q529=+0 ;MACHINING OPERATION ~	
Q221=+0 ;SURFACE OVERSIZE ~	
Q441=+1 ;INFEED ~	
Q449=+15000 ;FEED RATE ~	
Q491=+15 ;CONTOUR START RADIUS ~	
Q357=+2 ;CLEARANCE TO SIDE ~	
Q445=+50 ;CLEARANCE HEIGHT ~	
Q592=+1 ;TYPE OF DIMENSION	
9 L Z+50 R0 FMAX M3	; Pre-position in the tool axis, spindle ON
10 L X+0 Y+0 R0 FMAX M99	; Pre-position in the working plane to the center of rotation, call the cycle
11 M30	; End of program
12 LBL 1	; LBL1 contains the contour
13 L Z+2 X+15	
14 L Z-5	
15 L Z-7 X+19	
16 RND R3	
17 L Z-15	
18 RND R2	
19 L X+27	

20 LBL 0	
21 END PGM 6 MM	

Example of hob milling

The following NC program uses Cycle **286 GEAR HOBBING**. This programming example shows how to machine an involute spline with module = 1 (deviating from DIN 3960).

Program sequence

- Tool call: Gear hob
- Start the turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **286**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM 7 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI58	
2 TOOL CALL "GEAR_HOB"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
*** - ...**	; Reset the coordinate system
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	; Constant surface speed OFF
7 M140 MB MAX	; Retract the tool
8 L A+0 R0 FMAX	; Set the rotary axis to 0
9 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
10 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-11 ;END POINT IN Z ~	
Q540=+1 ;MODULE ~	
Q541=+90 ;NUMBER OF TEETH ~	
Q542=+90 ;OUTSIDE DIAMETER ~	
Q563=+1 ;TOOTH HEIGHT ~	
Q543=+0.05 ;TROUGH-TIP CLEARANCE ~	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 286 GEAR HOBBING ~	
Q215=+0 ;MACHINING OPERATION ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+30 ;CLEARANCE HEIGHT ~	
Q545=+1.6 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	
Q253=+2222 ;F PRE-POSITIONING ~	

Q553=+5 ;TOOL LENGTH OFFSET ~	
Q554=+10 ;SYNCHRONOUS SHIFT ~	
Q548=+1 ;ROUGHING SHIFT ~	
Q463=+1 ;MAX. CUTTING DEPTH ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	
Q478=+0.3 ;PLUNGING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q549=+3 ;FINISHING SHIFT	
13 CYCL CALL M303	; Call the cycle, spindle ON
14 FUNCTION MODE MILL	; Activate milling mode
15 M140 MB MAX	; Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	; Reset the rotation
17 M30	; End of program
18 END PGM 7 MM	

Example of skiving

The following NC program uses Cycle **287 GEAR SKIVING**. This programming example shows how to machine an involute spline with module = 1 (deviating from DIN 3960).

Program sequence

- Tool call: Internal gear cutter
- Start turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **287**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM 7 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI58	
2 TOOL CALL "SKIVING"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST: OFF S50	; Constant surface speed OFF
7 M140 MB MAX	; Retract the tool
8 L A+0 R0 FMAX	; Set the rotary axis to 0
9 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
10 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-11 ;END POINT IN Z ~	
Q540=+1 ;MODULE ~	
Q541=+90 ;NUMBER OF TEETH ~	
Q542=+90 ;OUTSIDE DIAMETER ~	
Q563=+1 ;TOOTH HEIGHT ~	
Q543=+0.05 ;TROUGH-TIP CLEARANCE ~	
Q544=+10 ;ANGLE OF INCLINATION	
12 CYCL DEF 287 GEAR SKIVING ~	
Q240=+5 ;CUTS/TABLE ~	
Q584=+1 ;NO. OF FIRST CUT ~	
Q585=+5 ;NO. OF LAST CUT ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+50 ;CLEARANCE HEIGHT ~	
Q545=+20 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	

Q253=+2222 ;F PRE-POSITIONING ~	
Q586=+0.4 ;FIRST INFEED ~	
Q587=+0.1 ;LAST INFEED ~	
Q588=+0.4 ;FIRST FEED RATE ~	
Q589=+0.25 ;LAST FEED RATE ~	
Q580=+0.2 ;FEED-RATE ADAPTION ~	
Q466=+2 ;OVERRUN PATH	
13 CYCL CALL M303	; Call the cycle, spindle ON
14 FUNCTION MODE MILL	; Activate milling mode
15 M140 MB MAX	; Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	; Reset the rotation
17 M30	; End of program
18 END PGM 7 MM	

Example of skiving with technology table and profile program

The NC program below uses Cycle **287 GEAR SKIVING** with the technology table. The technology table defines an individual tooth flank profile with symmetrical crowning for the last cut.

The profile program checks the defined machining side **Q550**, and the suitable infeed direction that matches this machining side is used.

Program sequence

- Tool call of a ring gear milling cutter
- Start the turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **287**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM SKIV MM	
1 BLK FORM CYLINDER Z R400 L20 DIST+0 DI300	
2 TOOL CALL "SKIVING"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST: OFF VC:200 S200	; Constant surface speed OFF
7 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
8 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
9 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-20 ;END POINT IN Z ~	
Q540=+4 ;MODULE ~	
Q541=-76 ;NUMBER OF TEETH ~	
Q542=+0 ;OUTSIDE DIAMETER ~	
Q563=+9 ;TOOTH HEIGHT ~	
Q543=+0 ;TROUGH-TIP CLEARANCE ~	
Q544=+0 ;ANGLE OF INCLINATION	
10 CYCL DEF 287 GEAR SKIVING ~	
QS240="SKIV.TAB" ;CUTS/TABLE ~	
Q584=+1 ;NO. OF FIRST CUT ~	
Q585=+99 ;NO. OF LAST CUT ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+50 ;CLEARANCE HEIGHT ~	
Q545=-20 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	

Q533=-1 ;PREFERRED DIRECTION ~	
Q530=+1 ;INCLINED MACHINING ~	
Q253=+2222 ;F PRE-POSITIONING ~	
Q586=+1.5 ;FIRST INFEED ~	
Q587=+0.1 ;LAST INFEED ~	
Q588=+2 ;FIRST FEED RATE ~	
Q589=+1 ;LAST FEED RATE ~	
Q580=+0.2 ;FEED-RATE ADAPTION ~	
Q466=+0.1 ;OVERRUN PATH	
11 L X+0 Y+0 R0 FMAX M136	
12 CYCL CALL M303	; Call the cycle, spindle ON
13 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
14 M305	
15 FUNCTION MODE MILL	; Activate milling mode
16 M140 MB MAX	; Retract the tool in the tool axis
17 L A+0 C+0 R0 FMAX	; Reset the rotation
18 M30	; End of program
19 END PGM SKIV MM	

Technology table SKIV.TAB

NR	FEED	INFEED	dY	dK	PGM
0	0.233	1.497	0	0	
1	0.251	1.265	0	0	
2	0.265	1.117	0	0	
3	0.278	1.01	0	0	
4	0.288	0.93	0	0.001	
5	0.298	0.866	0	-0.001	
6	0.307	0.813	0.01	0	
7	0.15	0.77	-0.01	0	
8	0.1	0.732	0	0	TNC:\Skiving\Prog_contour.h

Profile program

0 BEGIN PGM PROG_CONTOUR MM	
1 QL0 = +0	; Z1
2 QL1 = +0.03	; Y1
3 QL2 = -10	; Z2
4 QL3 = +0	; Y2
5 QL4 = -20	; Z3
6 QL5 = +0.03	; Y3
8 FN 9: IF Q550 EQU +0 GOTO LBL "machSideNeg"	; Selection of machining side
9 FN 23: QL10 = CDATA QL0	; Circle data from three points on the circle, QL10 = Circle center Z; QL11 = Circle center X; QL12 = Circle radius
10 L YQL1 ZQL0	
11 CR YQL5 ZQL4 RQL12 DR+	
12 FN 9: IF +0 EQU +0 GOTO LBL "END"	
13 LBL "machSideNeg"	
14 QL1 = -QL1	
15 QL3 = -QL3	
16 QL5 = -QL5	
17 FN 23: QL10 = CDATA QL0	; Circle data from three points on the circle
18 L YQL1 ZQL0	
19 CR YQL5 ZQL4 RQL12 DR-	
20 LBL "END"	
21 END PGM PROG_CONTOUR MM	

14

Cycles: Turning

14.1 Turning cycles (option 50)

Overview

To define turning cycles:

CYCL DEF

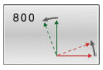
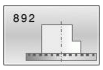
- Press the **CYCL DEF** key

TURNING

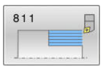
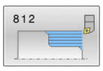
- Press the **TURNING** soft key
- Select cycle group (e.g., cycles for longitudinal turning)
- Select cycle (e.g., **SHOULDER, LONGITDNL.**)

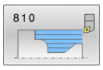
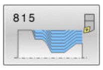
The control offers the following cycles for turning operations:

Special cycles

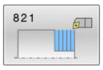
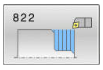
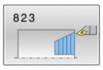
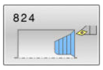
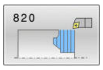
Soft key	Cycle	Page
	Cycle 800 ADJUST XZ SYSTEM ■ Moving the tool to a suitable position relative to the turning spindle	541
	Cycle 801 RESET ROTARY COORDINATE SYSTEM ■ Resetting Cycle **800**	549
	Cycle 880 GEAR HOBBING (option 50, option 131) ■ Description of the geometry and the tool ■ Selection of machining strategy and machining side	551
	Cycle 892 CHECK UNBALANCE (option 50) ■ Checking the unbalance of the turning spindle	560

Cycles for longitudinal turning

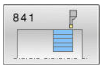
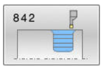
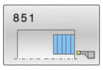
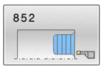
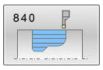
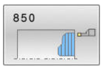
Soft key	Cycle	Page
	Cycle 811 SHOULDER, LONGITDNL. ■ Longitudinal turning of rectangular shoulders	565
	Cycle 812 SHOULDER, LONG. EXT. ■ Longitudinal turning of rectangular shoulders ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	569
	Cycle 813 TURN PLUNGE CONTOUR LONGITUDINAL ■ Longitudinal turning of shoulders with plunging elements	574

Soft key	Cycle	Page
	Cycle 814 TURN PLUNGE LONGITUDINAL EXT. ■ Longitudinal turning of shoulders with plunging elements ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	578
	Cycle 810 TURN CONTOUR LONG. ■ Longitudinal turning of turning contours of any shape ■ Removing stock paraxially	583
	Cycle 815 CONTOUR-PAR. TURNING ■ Longitudinal turning of turning contours of any shape ■ Removing of stock is performed parallel to the contour	588

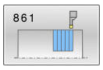
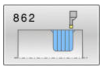
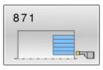
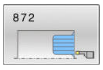
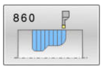
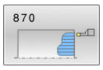
Cycles for transverse turning

Soft key	Cycle	Page
	Cycle 821 SHOULDER, FACE ■ Face turning of rectangular shoulders	592
	Cycle 822 SHOULDER, FACE. EXT. ■ Face turning of rectangular shoulders ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	596
	Cycle 823 TURN TRANSVERSE PLUNGE ■ Face turning of shoulders with plunging elements	601
	Cycle 824 TURN PLUNGE TRANSVERSE EXT. ■ Face turning of shoulders with plunging elements ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	605
	Cycle 820 TURN CONTOUR TRANSV. ■ Face turning of turning contours of any shape	610

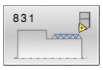
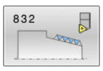
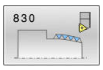
Cycles for recess turning

Soft key	Cycle	Page
	Cycle 841 SIMPLE REC. TURNG., RADIAL DIR. ■ Recess turning of rectangular slots in longitudinal direction	615
	Cycle 842 ENH.REC.TURNNG, RAD. ■ Recess turning of slots in longitudinal direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	619
	Cycle 851 SIMPLE REC TURNG, AX ■ Recess turning of slots in transverse direction	624
	Cycle 852 ENH.REC.TURNING, AX. ■ Recess turning of slots in transverse direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	628
	Cycle 840 RECESS TURNG, RADIAL ■ Recess turning of slots of any shape in longitudinal direction	633
	Cycle 850 RECESS TURNG, AXIAL ■ Recess turning of slots of any shape in transverse direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	638

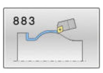
Cycles for recessing

Soft key	Cycle	Page
	Cycle 861 SIMPLE RECESS, RADL. ■ Radial recessing of rectangular slots	643
	Cycle 862 EXPND. RECESS, RADL. ■ Radial recessing of rectangular slots ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	648
	Cycle 871 SIMPLE RECESS, AXIAL ■ Axial recessing of rectangular slots	654
	Cycle 872 EXPND. RECESS, AXIAL ■ Axial recessing of rectangular slots ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	659
	Cycle 860 CONT. RECESS, RADIAL ■ Radial recessing of slots of any shape	666
	Cycle 870 CONT. RECESS, AXIAL ■ Axial recessing of slots of any shape	672

Cycles for thread turning

Soft key	Cycle	Page
	Cycle 831 THREAD LONGITUDINAL ■ Longitudinal turning of threads	678
	Cycle 832 THREAD EXTENDED ■ Longitudinal or face turning of threads and tapered threads ■ Definition of an approach path and an idle travel path	683
	Cycle 830 THREAD CONTOUR-PARALLEL ■ Longitudinal or face turning of threads of any shape ■ Definition of an approach path and an idle travel path	688

Advanced turning functions

Soft key	Cycle	Page
	Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option 158) ■ Roughing of complex contours with different angles of inclination	694
	Cycle 883 TURNING SIMULTANEOUS FINISHING (option 158) ■ Roughing of complex contours with different angles of inclination	701

Working with turning cycles

In turning cycles, the control takes the cutting geometry (**TO, RS, P-ANGLE, T-ANGLE**) of the tool into account in order to prevent damage to the defined contour elements. If it is not possible to machine the entire contour with the active tool, the control will display a warning.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the control detects the machining position (inside or outside machining) via the starting position or tool position when the cycle is called. In some cycles you can also enter the machining position directly in the cycle. After modifying the machining position, check the tool position and the direction of rotation.

If you program **M136** before a cycle, the control interprets feed rate values in the cycle in mm/rev.; without **M136** in mm/min.

If you execute turning cycles with inclined machining (**M144**), the angles of the tool with respect to the contour change. The control automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

Some cycles machine contours that you have written in a subprogram. You can program these contours with Klartext contouring functions or FK functions. Before calling the cycle, you must program the cycle **14 CONTOUR** to define the subprogram number.

The turning cycles 81x - 87x as well 880, 882, and 883 must be called with **CYCL CALL** or **M99**. Before programming a cycle call, be sure to program:

- Turning mode: **FUNCTION MODE TURN**
- Call a tool with **TOOL CALL**
- Direction of rotation of turning spindle (e.g., **M303**)
- Selection of speed or cutting speed: **FUNCTION TURNDATA SPIN**
- If you use feed rate per revolution mm/rev., **M136**
- Position the tool to a suitable starting point (e.g., **L X+130 Y +0 R0 FMAX**)
- Adapt the coordinate system, and align the tool: **CYCL DEF 800 ADJUST XZ SYSTEM**

> **i**
>
> - If the control is unable to machine the entire contour in turning cycles, it will display locations with residual material in the simulation. The control displays the tool path in yellow instead of white and crosshatches the residual material.
> - The control will always display yellow tool paths and the crosshatching, independent of the selected mode, model quality, and display mode of the tool paths.
> - The control does not use the **BLK FORM** function to generate the traverse paths for the turning cycles (option 50). Define **FUNCTION TURNDATA BLANK**
>
>   **Further information:** User's Manual for Programming and Testing

Recesses and undercuts

Application

Some cycles machine contours that you have written in a subprogram. Further special contour elements are available to you for writing turning contours. In this way you can program recessing and undercutting as complete contour elements with a single NC block.

> Recessing and undercutting are always referenced to a previously defined linear contour element.

Description of function

Various input options are available to you for defining undercuts and recesses. Some of these inputs have to be made (mandatory input); others can be skipped (optional input). The mandatory inputs are symbolized as such in the help graphics. In some elements, you can select between two different definitions. The control provides soft keys with the corresponding selection possibilities.

Programming recessing and undercutting:

SPEC FCT

- Press the **SPEC FCT** key

TURNING PROGRAM FUNCTIONS

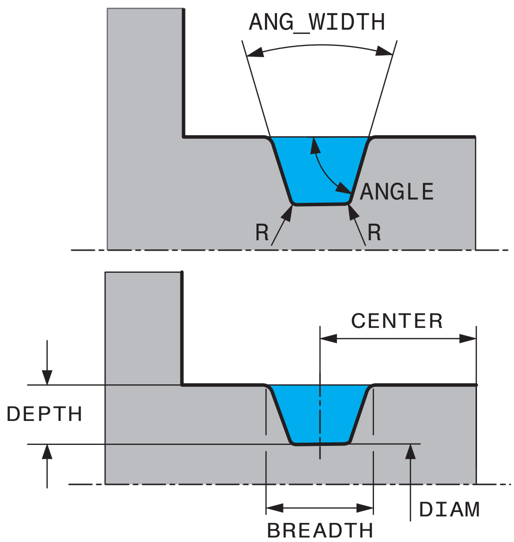
- Press the **TURNING PROGRAM FUNCTIONS** soft key
-

RECESS/ UNDERCUT

- Press the **RECESS/ UNDERCUT** soft key

GRV

- Press the **GRV** (recess) or **UDC** (undercut) soft key



Programming recessing

Recessing is the machining of recesses into round parts, usually for accommodation of locking rings and seals, or as lubricating grooves. You can program recessing around the circumference or on the face end of the turned part. You have two separate contour elements for this purpose:

- **GRV RADIAL**: Recess in circumference of component
- **GRV AXIAL**: Recess on face end of component

Input parameters in recessing GRV

Parameter	Meaning	Input
CENTER	Center of recess	Required
R	Corner radius of both inside corners	Optional
DEPTH / **DIAM**	Depth of recess (pay attention to algebraic sign!) /diameter of recess base	Required
BREADTH	Recess width	Required
ANGLE / **ANG_WIDTH**	Flank angle / opening angle between both flanks	Optional
RND / **CHF**	Rounding / chamfer on contour corner near to starting point	Optional
FAR_RND / **FAR_CHF**	Rounding / chamfer on contour corner away from starting point	Optional

The algebraic sign for the recess depth specifies the machining position (inside/outside machining) of the recess.

Algebraic signs of recess depth for outside machining:

- If the contour element is in the negative direction of the Z coordinate, use a negative sign
- If the contour element is in the positive direction of the Z coordinate, use a positive sign

Algebraic signs of recess depth for inside machining:

- If the contour element is in the negative direction of the Z coordinate, use a positive sign
- If the contour element is in the positive direction of the Z coordinate, use a negative sign

Example: Radial recess with depth = 5, width = 10, pos. = Z-15

```
11 L  X+40  Z+0
12 L  Z-30
13 GRV RADIAL CENTER-15 DEPTH-5 BREADTH10 CHF1 FAR_CHF1
14 L  X+60
```

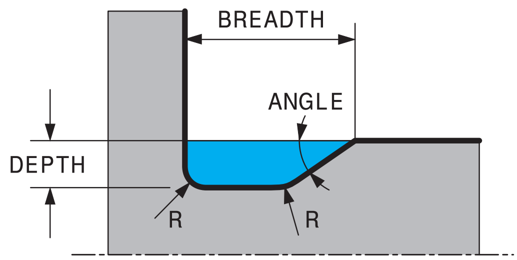
Programming undercutting

Undercutting is usually required for the flush connection of components. In addition, undercutting can help reduce the notch effect at corners. Threads and fits are often machined with an undercut. You have various contour elements for defining the different undercuts:

- **UDC TYPE_E**: Undercut for cylindrical surfaces to be further processed as per DIN 509.
- **UDC TYPE_F**: Undercut for plane surface and cylindrical surface to be further processed as per DIN 509
- **UDC TYPE_H**: Undercut for more rounded transition as per DIN 509
- **UDC TYPE_K**: Undercut in plane surface and cylindrical surface
- **UDC TYPE_U**: Undercut in cylindrical surface
- **UDC THREAD**: Thread undercut as per DIN 76

> The control always interprets undercuts as form elements in the longitudinal direction. No undercuts are possible in the plane direction.

Undercut DIN 509 UDC TYPE _E



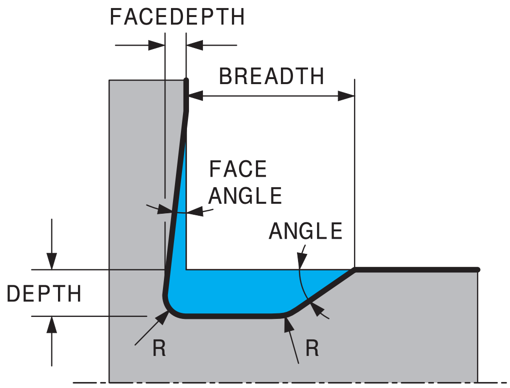
Input parameters in undercut DIN 509 UDC TYPE_E

Parameter	Meaning	Input
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Undercut with depth = 2, width = 15

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_E R1 DEPTH2 BREADTH15
14 L  X+60
```

Undercut DIN 509 UDC TYPE_F



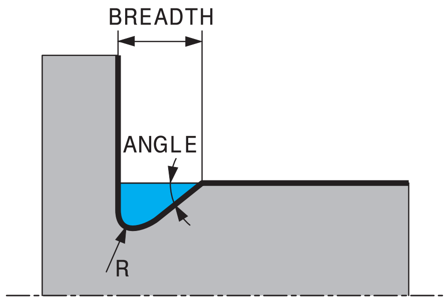
Input parameters in undercut DIN 509 UDC TYPE_F

Parameter	Meaning	Input
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional
FACEDEPTH	Depth of face	Optional
FACEANGLE	Contour angle of face	Optional

Example: Undercut form F with depth = 2, width = 15, depth of face = 1

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_F R1 DEPTH2 BREADTH15 FACEDEPTH1
14 L  X+60
```

Undercut DIN 509 UDC TYPE_H



Input parameters in undercut DIN 509 UDC TYPE_H

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
BREADTH	Width of undercut	Required
ANGLE	Undercut angle	Required

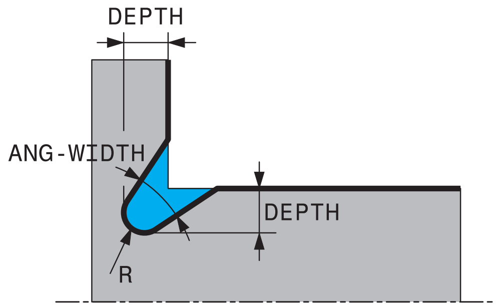
Example: Undercut form H with depth = 2, width = 15, angle = 10°

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_H R1 BREADTH10 ANGLE10
14 L  X+60
```

Undercut UDC TYPE_K

Input parameters in undercut UDC TYPE_K

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
DEPTH	Undercut depth (parallel to axis)	Required
ROT	Angle relative to longitudinal axis (default: 45°)	Optional
ANG_WIDTH	Angle of undercut opening	Required



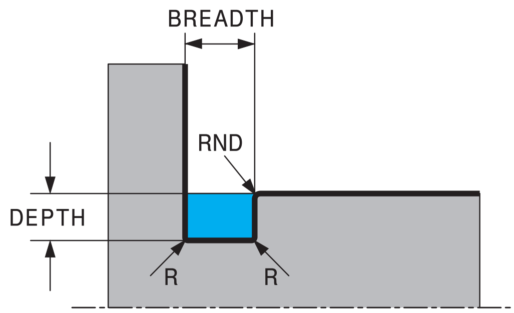
Example: Undercut form K with depth = 2, width = 15, opening angle = 30°

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_K R1 DEPTH3 ANG_WIDTH30
14 L  X+60
```

Undercut UDC TYPE_U

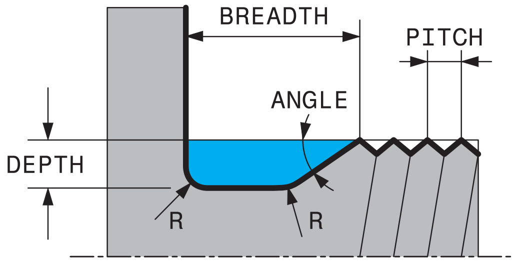
Input parameters in undercut UDC TYPE_U

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
DEPTH	Undercut depth	Required
BREADTH	Width of undercut	Required
RND / CHF	Rounding / chamfer on outside corner	Required



Example: Undercut form U with depth = 3, width = 8

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_U R1 DEPTH3 BREADTH8 RND1
14 L  X+60
```



Undercut UDC THREAD

Input parameters in undercut DIN 76 UDC THREAD

Parameter	Meaning	Input
PITCH	Thread pitch	Optional
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Thread undercut according to DIN 76 with thread pitch = 2

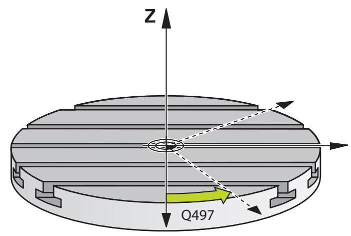
```
11 L  X+40  Z+0
12 L  Z-30
13 UDC THREAD PITCH2
14 L  X+60
```

14.2 Cycle 800 ADJUST XZ SYSTEM

ISO programming

G800

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
> The cycle is machine-dependent.

To be able to perform a turning operation, you need to position the tool appropriately relative to the workspindle. For this purpose, you can use Cycle **800 ADJUST XZ SYSTEM**.

With turning operations, the inclination angle between the tool and workspindle is important, for example to machine contours with undercuts. Cycle **800** provides various options for aligning the coordinate system for an inclined machining operation:

- If you have positioned the tilting axis for inclined machining, you can use Cycle **800** to align the coordinate system relative to the positions of the tilting axes (**Q530 = 0**). In this case, make sure to program **M144** or **M128**/**TCPM** for proper calculation of the alignment
- Cycle **800** calculates the required tilting axis angle based on the inclination angle **Q531** – depending on the strategy selected in the **INCLINED MACHINING Q530** parameter, the control positions the tilting axis with (**Q530 = 1**) or without compensation movement (**Q530 = 2**)
- Cycle **800** uses the inclination angle **Q531** to calculate the required tilting axis angle, but does not position the tilting axis (**Q530 = 3**). You need to position the tilting axis manually to the calculated values **Q120** (A axis), **Q121** (B axis), and **Q122** (C axis) after the cycle

If the milling spindle axis and the workspindle axis are parallel to each other, you can use the **Precession angle Q497** to define any desired rotation of the coordinate system about the spindle axis (Z axis). This may be necessary if you have to bring the tool into a specific position due to a lack of space or if you want to be able to optimally monitor a machining process. If the axes of the workspindle and of the milling spindle are not parallel, only two precession angles are realistic for machining. The control selects the angle that is closest to the input value of **Q497**.

Cycle **800** positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can use a mirrored version of the tool (**REVERSE TOOL Q498**); this offsets the milling spindle by 180°. In this way, you can use your tools for both internal and external machining. Position the cutting edge at the center of the workspindle by using a positioning block, such as **L Y+0 R0 FMAX**.

- If you change the position of a tilting axis, you need to run Cycle **800** again to align the coordinate system.
- Check the orientation of the tool before machining.

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of the center of rotation is aligned with the axis of the workspindle. For example, this is the case with large or non-rotationally symmetrical workpieces. The eccentric turning **Q535** function in Cycle **800** enables you to perform turning in such cases as well.

During eccentric turning, more than one linear axis is coupled to the workspindle. The control compensates the eccentricity by performing circular compensation movements with the coupled linear axes.

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

If you machine at high spindle speed and with a high amount of eccentricity, you need to program large feed rates for the linear axes in order to perform the movements synchronously. If these feed rates cannot be met, the contour will be damaged. The control therefore generates an error message if 80% of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the spindle speed.

Operating notes

NOTICE
Danger of collision!
The control performs compensating movements during coupling and decoupling. There is a danger of collision!
▶ Coupling and decoupling must be performed while the spindle is stationary

NOTICE
Caution: Danger to the tool and workpiece!
The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.
▶ Select the technology data in such a way that no vibrations (resonances) occur

- Turn a test cut before the actual machining operation to ensure that the required speeds can be attained.
- The linear axis positions resulting from the compensation are displayed by the control only in the ACTUAL value position display.

Effect

With Cycle **800 ADJUST XZ SYSTEM**, the control aligns the workpiece coordinate system and orients the tool correspondingly. Cycle **800** is effective until it is reset by Cycle **801**, or until Cycle **800** is redefined. Some cycle functions of Cycle **800** are implicitly reset by other factors:

- Mirroring of tool data (**Q498 REVERSE TOOL**) is reset by a tool call with **TOOL CALL**
- The **ECCENTRIC TURNING Q535** function is reset at the end of the program or if the program is aborted (internal stop)

Notes

The machine manufacturer configures your machine tool. If the tool spindle was defined as an axis in the kinematic model during this configuration, the feed-rate potentiometer is effective for movements related to Cycle **800**.

The machine manufacturer can configure a grid for the positioning of the tool spindle.

NOTICE

Danger of collision!

If the milling spindle was defined as an NC axis in turning mode, then the control is able to derive a tool reversal from the axis position. However, if the milling spindle was defined as a spindle, there is a risk that the tool reversal definition might get lost! There is a danger of collision!

- Enable tool reversal again after a **TOOL CALL** block

NOTICE

Danger of collision!

If **Q498** = 1 and you additionally program the **FUNCTION LIFTOFF ANGLE TCS** function, then there might be two different results, depending on the configuration. If the tool spindle has been defined as an axis, the **LIFTOFF** will be included in the rotation during tool reversal. If the tool spindle has been defined as a kinematic transformation, then the **LIFTOFF** will **not** be included in the rotation during tool reversal! There is a danger of collision!

- Carefully test the NC program or program section in **Program run, single block** operating mode
- If required, change the algebraic sign of the SPB angle.

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool must be clamped and measured in the correct position.
- Cycle **800** positions only the first rotary axis based on the tool position. If an **M138** is activated, then this limits the selection to the defined rotary axes. If you want to move other rotary axes to a specific position, then position theses axes correspondingly before running Cycle **800**.

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

Notes on programming

- You can mirror the tool data (**Q498 REVERSE TOOL**) only if a turning tool has been selected.
- To reset Cycle **800**, program Cycle **801 RESET ROTARY COORDINATE SYSTEM**.
- Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine manufacturer) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.
- If the workpiece is to be rotated about the workpiece spindle, then use an offset of the workpiece spindle in the preset table. Basic rotations are not permitted; the control issues an error message.
- If you set parameter **Q530** "Inclined machining" to 0 (tilting axes must have been positioned previously), make sure to program **M144** or **TCPM/M128** beforehand.
- If, in parameter **Q530** “Inclined machining,” you use the settings 1: MOVE, 2: TURN and 3: STAY, then the control, depending on the machine configuration, activates function **M144** or TCPM (**Further information:** Setup, Testing and Running NC Programs User’s Manual)

Cycle parameters

Help graphic	Parameters
	Q497 Precession angle? Angle at which the control positions the tool. Input: **0.00000...359.99999**
	Q498 Reverse tool (0=no/1=yes)? Mirror tool for inside/outside machining. Input: **0**, **1**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **0**: Maintain tilting axis position (axis must be positioned beforehand) **1**: Automatically position the tilting axis, and orient the tool tip (MOVE). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (TURN) **3**: Do not position the tilting axis. Position the tilting axes later in a separate positioning block (STAY). The control stores the position values in the parameters **Q120** (A axis), **Q121** (B axis) and **Q122** (C axis). Input: **0**, **1**, **2**, **3**
	Q531 Angle of incidence? Angle of incidence for positioning the tool Input: **-180.00000...+180.00000**
	Q532 Feed rate for positioning? Traversing speed of the tilting axis during automatic positioning Input: **0.001...99999.999** or **FMAX**
	Q533 Preferred dir. of incid. angle? **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**

Help graphic	Parameters
	Q535 Eccentric turning? Couple the axes for the eccentric turning operation: **0**: Deactivate axis couplings **1**: Activate axis couplings. The center of rotation is located at the active preset **2**: Activate axis couplings. The center of rotation is located at the active datum **3**: Do not change the axis couplings Input: **0**, **1**, **2**, **3**
	Q536 Eccentric turning without stop? Interrupt program run before the axes are coupled: **0**: Stop before the axes are coupled again. In stopped condition, the control opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. You can then continue the machining operating with **NC-Start** or select **CANCEL** **1**: Axes are coupled without stopping beforehand Input: **0**, **1**
	Q599 or **QS599 Retraction path/macro?** Retraction prior to execution of positioning movements in the rotary axis or tool axis: **0**: No retraction **-1**: Maximum retraction with **M140 MB MAX** **Further information:** Programming and Testing User's Manual **> 0**: Path for the retraction in **mm** or **inches** **"..."**: Path for an NC program that will be called as a user macro. **Further information:** "User macro", Page 548 Input: **-1...9999** in the case of text entry: maximum **255** characters or **QS** parameter

Example

```
11 CYCL DEF 800 ADJUST XZ SYSTEM ~
    Q497=+0                 ;PRECESSION ANGLE ~
    Q498=+0                 ;REVERSE TOOL ~
    Q530=+0                 ;INCLINED MACHINING ~
    Q531=+0                 ;ANGLE OF INCIDENCE ~
    Q532=+750               ;FEED RATE ~
    Q533=+0                 ;PREFERRED DIRECTION ~
    Q535=+3                 ;ECCENTRIC TURNING ~
    Q536=+0                 ;ECCENTRIC W/O STOP ~
    Q599=-1                 ;RETRACT
```

User macro

A user macro is another NC program.

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

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

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

  Further information: Klartext Programming User's Manual

Example of a user macro for retraction

0 BEGIN PGM RET MM	
1 FUNCTION RESET TCPM	; Reset TCPM
2 L Z-1 R0 FMAX M91	; Traverse with M91
3 FN 10: IF Q533 NE+0 GOTO LBL "DEF_DIRECTION"	; If Q533 (preferred direction from Cycle 800) is not equal to 0, then jump to LBL "DEF_DIRECTION"
4 FN 18: SYSREAD QL1 = ID240 NR1 IDX4	; Read system data (nominal position in the REF system) and store in QL1
5 QL0 = 500 * SGN QL1	; SGN = Check algebraic sign
6 FN 9: IF +0 EQU +0 GOTO LBL "MOVE"	; Jump to LBL MOVE
7 LBL "DIRECTION"	
8 QL0 = 500 * SGN Q533	; SGN = Check algebraic sign
9 LBL "MOVE"	
10 L X-500 Y+QL0 R0 FMAX M91	; Retraction with M91
11 END PGM RET MM	

14.3 Cycle 801 RESET ROTARY COORDINATE SYSTEM

ISO programming
G801

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
> The cycle is machine-dependent.

Cycle **801** resets the following settings you have programmed with Cycle **800**:

- Precession angle **Q497**
- Reverse tool **Q498**

If you executed the eccentric turning function with Cycle **800**, please note the following: Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine manufacturer) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.

> Cycle **801** does not orient the tool to the starting position. If a tool was oriented with Cycle **800**, it remains in this position also after resetting.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- With Cycle **801 RESET ROTARY COORDINATE SYSTEM**, you can reset the settings you have made with Cycle **800 ADJUST XZ SYSTEM**.
- Cycle **801** does not result in any axis movement. To bring an inclined axis into home position, program Cycle **800 ADJUST XZ SYSTEM** with **Q531 ANGLE OF INCIDENCE** equal to **0** or **PLANE RESET**.

Notes on programming

- Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine manufacturer) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.

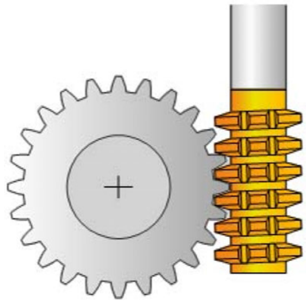
Cycle parameters

Help graphic	Parameter
	Cycle **801** does not have a cycle parameter. Close cycle input with the **END** key.

14.4 Cycle 880 GEAR HOBBING (option 50, option 131)

ISO programming
G880

Application



Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.

With Cycle **880 GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. In the cycle you first define the **gear** and then the **tool** with which the gear is to be machined. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary motion of the tool spindle and rotary table. In addition, the gear hob moves along the workpiece in axial direction.

While Cycle **880 GEAR HOBBING** is active, the coordinate system might be rotated. It is therefore essential to program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145** after the end of the cycle.

Related topics

- Cycle **286 GEAR HOBBING**

  Further information: "Cycle 286 GEAR HOBBING (option 157)", Page 488

Cycle run

1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate FMAX. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the FMAX feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate **Q253**; **M144** is internally active in the cycle
4 The control positions the tool at the feed rate FMAX to the starting point in the working plane.
5 The control then moves the tool in the tool axis at the feed rate **Q253** to set-up clearance **Q460**.
6 The control now moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q460** and the end point in Z **Q552**+**Q460**.
7 When the control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX
10 The machining operation ends in the tilted system.
11 Now you need to move the tool to a safe height and reset the tilting of the working plane.
12 It is essential that you now program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145**

Notes

NOTICE

Danger of collision!

If you do not position the tool to a safe position, a collision may occur between the tool and workpiece (fixtures) during tilting.

- Pre-position the tool so that it is already on the desired machining side **Q550**.
- Move the tool to a safe position on this machining side

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q460**!

- Clamp the workpiece out of the fixtures far enough to prevent a danger of collision between the tool and the fixtures
- Clamp the workpiece in such a way that its protrusion from the fixture will not cause any collision when the tool is automatically moved to the starting or end point using a path that is extended by the set-up clearance **Q460**

NOTICE

Danger of collision!

Depending on whether you use **M136** or not, the feed rate values will be interpreted differently by the control. If the programmed feed rate was too high, the workpiece might be damaged.

- If you program **M136** explicitly before the cycle, the control will interpret the feed rates in the cycle in mm/rev.
- If you do not program **M136** before the cycle, the control will interpret the feed rates in the cycle in mm/min.

NOTICE

Danger of collision!

If you do not reset the coordinate system after Cycle **880**, the precession angle set by the cycle will remain active. There is a danger of collision!

- Make sure to program Cycle **801** after Cycle **880** in order to reset the coordinate system.
- Make sure to program Cycle **801** after a program abort in order to reset the coordinate system.

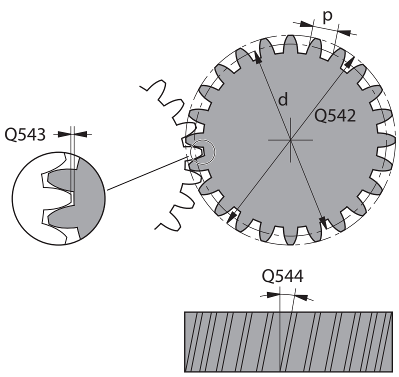
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- Define the tool as a milling cutter in the tool table.
- Before programming the cycle call, set the datum to the center of rotation.

> So as to avoid exceeding the maximum permissible spindle speed of the tool, you can program a limitation. (Specify it in the **Nmax** column of the "tool.t" tool table.)

Notes on programming

- The values entered for the module, number of teeth and outside diameter (outside diameter) are monitored. If these values are not coherent, then an error message is displayed. You can fill in 2 of the 3 parameters. Enter 0 for the module, the number of teeth, or the outside diameter (outside diameter). In this case, the control will calculate the missing value.
- Program FUNCTION TURNDATA SPIN VCONST:OFF.
- If you program FUNCTION TURNDATA SPIN VCONST:OFF S15, then the spindle speed of the tool is calculated as follows: **Q541** x S. With **Q541**=238 and S=15, this would result in a tool spindle speed of 3570 rpm.
- Program the direction of rotation of your workpiece (**M303**/**M304**) before the start of the cycle.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q540 Module? Module of the gear Input: **0...99.999**
	Q541 Number of teeth? Describe gear: number of teeth Input: **0...99999**
	**Q542 Outside diameter?** Describe gear: outside diameter of finished part Input: **0...99999.9999**
	Q543 Trough-to-tip clearance? Distance between the addendum circle of the gear to be made and root circle of the mating gear. Input: **0...9.9999**
	Q544 Angle of inclination? Angle at which the teeth of a helical gear are inclined relative to the direction of the axis. For straight-cut gears, this angle is 0°. Input: **-60...+60**
	Q545 Tool lead angle? Angle of the edges of the gear hob. Enter this value in decimal notation. Example: 0°47'=0.7833 Input: **-60...+60**
	Q546 Reverse tool rotation direction? Describe tool: Direction of spindle rotation of the gear hob **3**: Clockwise rotating tool (**M3**) **4**: Counterclockwise rotating tool (**M4**) Input: **3**, **4**
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**

Help graphic	Parameter
	Q550 Machining side (0=pos./1=neg.)? Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	Q253 Feed rate for pre-positioning? Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	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**
	Q553 TOOL:L offset, machining start? Define the minimum length offset (L OFFSET) that the tool should have when in use. The control offsets the tool in the longitudinal direction by this amount. This value has an incremental effect. Input: **0...999.999**

Help graphic	Parameter
	Q551 Starting point in Z? Starting point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
	Q552 End point in Z? End point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0.001...999.999**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q488 Feed rate for plunging Feed rate of the tool infeed Input: **0...99999.999** or **FAUTO**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

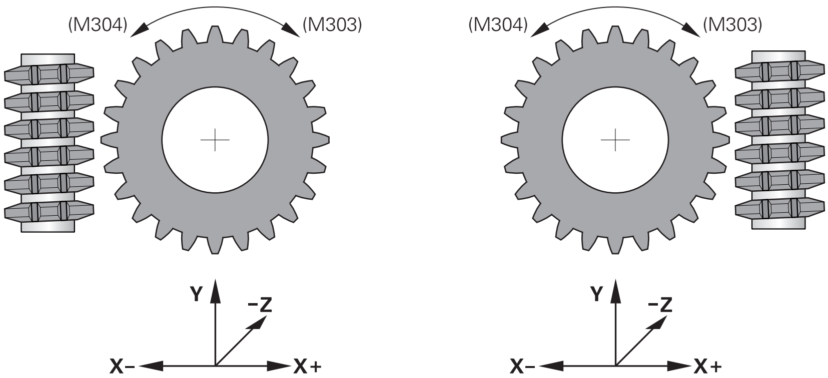
Example

```
11 CYCL DEF 880 GEAR HOBBING ~
    Q215=+0           ;MACHINING OPERATION ~
    Q540=+0           ;MODULE ~
    Q541=+0           ;NUMBER OF TEETH ~
    Q542=+0           ;OUTSIDE DIAMETER ~
    Q543=+0.1666      ;TROUGH-TIP CLEARANCE ~
    Q544=+0           ;ANGLE OF INCLINATION ~
    Q545=+0           ;TOOL LEAD ANGLE ~
    Q546=+3           ;CHANGE TOOL DIRECTN. ~
    Q547=+0           ;ANG. OFFSET, SPINDLE ~
    Q550=+1           ;MACHINING SIDE ~
    Q533=+0           ;PREFERRED DIRECTION ~
    Q530=+2           ;INCLINED MACHINING ~
    Q253=+750         ;F PRE-POSITIONING ~
    Q260=+100         ;CLEARANCE HEIGHT ~
    Q553=+10          ;TOOL LENGTH OFFSET ~
    Q551=+0           ;STARTING POINT IN Z
    Q552=-10          ;END POINT IN Z
    Q463=+1           ;MAX. CUTTING DEPTH ~
    Q460=+2           ;SAFETY CLEARANCE ~
    Q488=+0.3         ;PLUNGING FEED RATE ~
    Q478=+0.3         ;ROUGHING FEED RATE ~
    Q483=+0.4         ;OVERSIZE FOR DIAMETER ~
    Q505=+0.2         ;FINISHING FEED RATE
```

Direction of rotation depending on the machining side (Q550)

Determine the direction of rotation of the rotary table:

1 **What tool? (Right-cutting/left-cutting?)**
2 **What machining side? X+ (Q550=0) / X- (Q550=1)**
3 **Look up the direction of rotation of the rotary table in one of the two tables below!** To do so, select the appropriate table for the direction of rotation of your tool (**right-cutting/left-cutting**). Please refer to the tables below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)** ab.



Tool: Right-cutting M3	
Machining side X+ (Q550=0)	Direction of rotation of the table: Clockwise (M303)
Machining side X- (Q550=1)	Direction of rotation of the table: Counterclockwise (M304)

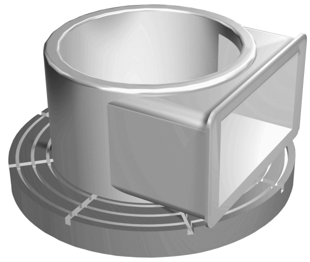
Tool: Left-cutting M4	
Machining side X+ (Q550=0)	Direction of rotation of the table: Counterclockwise (M304)
Machining side X- (Q550=1)	Direction of rotation of the table: Clockwise (M303)

14.5 Cycle 892 CHECK UNBALANCE (option 50)

ISO programming
G892

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



An unbalance can occur when turning an unsymmetrical workpiece, such as a pump body. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle **892 CHECK UNBALANCE**, the control checks the unbalance of the turning spindle. This cycle uses two parameters. **Q450** describes the maximum unbalance and **Q451** the maximum spindle speed. **If the maximum unbalance is exceeded, an error message is displayed and the NC program is aborted.** If the maximum unbalance is not exceeded, the control executes the NC program without interruption. This function protects the machine mechanics. It enables you to take action if an excessive unbalance is detected.

Notes

Cycle **892 CHECK UNBALANCE** can be hidden with the optional machine parameter **hideUnbalance** (no. 128902).

Your machine manufacturer configures Cycle **892**.

Your machine manufacturer defines the function of Cycle **892**.

The turning spindle rotates during the unbalance check.

This function can also be run on machines with more than one turning spindle. Contact the machine manufacturer for further information.

You need to check the applicability of the control's internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.

NOTICE

Danger of collision!

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance. If high unbalance loads are not compensated for, then this may lead to defects on the machine.

- Before starting a new machining cycle, run Cycle **892**.
- If required, use balancing weights to compensate for any unbalance.

NOTICE

Danger of collision!

The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps. If high unbalance loads are not compensated, then this may lead to defects on the machine

- Make sure to also run Cycle **892** between the machining steps.
- If required, use balancing weights to compensate for any unbalance.

NOTICE

Danger of collision!

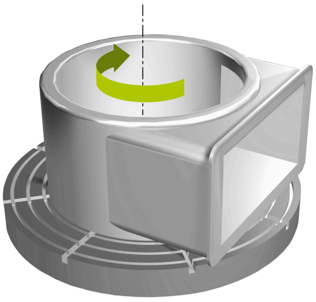
High unbalance loads, especially in combination with a high mass, may damage the machine. Consider the mass and unbalance of the workpiece when choosing the speed.

- Do not program high speeds with heavy workpieces or high unbalance loads.

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- If Cycle **892 CHECK UNBALANCE** has aborted the NC program, then we recommend that you use the manual MEASURE UNBALANCE cycle. With this cycle, the control determines the unbalance and calculates the mass and position of a balancing weight.

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

Cycle parameters

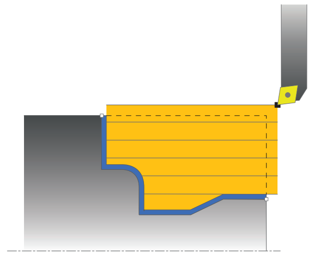
Help graphic	Parameter
	**Q450 Max. permissible runout?** Specifies the maximum runout of a sinusoidal unbalance signal in millimeters (mm). The signal results from the following error of the measuring axis and from the spindle revolutions. Input: **0...99999.9999**
	Q451 Rotational speed? Enter the rotational speed in revolutions per minute. The test for an unbalance begins with a low initial speed (e.g., 50 rpm). It is then automatically increased by specified increments (e.g., 25 rpm). until the maximum speed defined in parameter **Q451** is reached. Spindle speed override is disabled. Input: **0...99999**

Example

```
11 CYCL DEF 892 CHECK UNBALANCE ~
    Q450=+0              ;MAXIMUM RUNOUT ~
    Q451=+50             ;SPEED
```

14.6 Fundamentals of turning cycles

Refer to your machine manual.
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
Option 50 must have been enabled.



The pre-positioning of the tool has a decisive influence on the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when the cycle is called. When calculating the area to be machined, the control takes into account the starting point and the end point defined in the cycle or of the contour defined in the cycle. If the starting point is within the area to be machined, then the control positions the tool at the set-up clearance beforehand in some cycles.

The direction of stock removal is longitudinal to the rotary axis for Cycles **81x** and transverse to the rotary axis for Cycles **82x**. In Cycle **815**, the movements are contour-parallel.

The cycles can be used for inside and outside machining. The control takes the information for this from the position of the tool or from the definition in the cycle.

Further information: "Working with turning cycles", Page 534

For cycles in which a defined contour is machined (Cycles **810**, **820**, and **815**), the direction set when programming the contour determines the machining direction.

In cycles for turning you can specify the machining strategies of roughing, finishing or complete machining.

NOTICE

Danger of collision!

The turning cycles position the tool automatically to the starting point during finishing. The approach strategy is influenced by the position of the tool when the cycle is called. The decisive factor is whether the tool is located inside or outside an envelope contour when the cycle is called. The envelope contour is the programmed contour, enlarged by the set-up clearance. If the tool is within the envelope contour, the cycle positions the tool at the defined feed rate directly to the starting position. This can cause contour damage.

- Position the tool at a sufficient distance from the starting point to prevent the possibility of contour damage
- If the tool is outside the envelope contour, positioning to the envelope contour is performed at rapid traverse, and at the programmed feed rate within the envelope contour.

The control monitors the length of the cutting edge **CUTLENGTH** in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the length of the cutting edge defined in the tool table, then the control issues a warning. In this case, the cutting depth will be reduced automatically in the machining cycle.

Execution with a FreeTurn tool

The control supports the execution of the contours with FreeTurn tools in the cycles **81x** and **82x**. This method allows you to perform the most common turning operation with just one tool. Thanks to the flexible tool, machining times can be reduced because the control does not need to change tools as much.

Requirements

- The tool must be correctly defined.

NOTICE

Danger of collision!

The shaft length of the turning tool limits the diameter that can be machined. There is a risk of collision during machining!

- Check the machining sequence in the simulation

- The NC program remains unchanged except for the calling of the FreeTurn cutting edges.

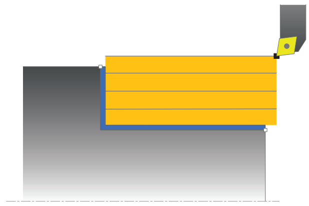
  Further information: "Example: Turning with a FreeTurn tool", Page 715
- If you use a FreeTurn tool for machining, the control will internally switch the kinematics. This can lead to movements changing the positions of the cutting edge. In this case, the control will display a warning message.

  If the control displays a warning message during simulation, HEIDENHAIN recommends that you run the program once without a workpiece. It is possible that the control does not display a warning during program run because the simulation does not show all movements, such as PLC positioning movements. The simulation may thus differ from the actual machining process.

14.7 Cycle 811 SHOULDER, LONGITDNL.

ISO programming
G811

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to carry out longitudinal turning of right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is execute paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The cycle processes the area from the tool position to the end point defined in the cycle.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. The control performs a paraxial infeed movement at rapid traverse.
3. The control finishes the contour of the finished part at the defined feed rate **Q505**.
4. The control retracts the tool at the defined feed rate to the set-up clearance.
5. The control returns the tool at rapid traverse to the cycle starting point.

Notes

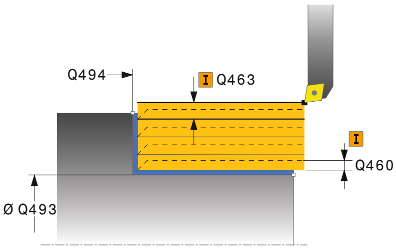
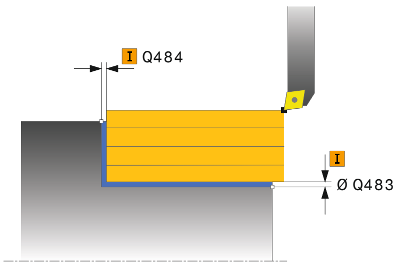
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

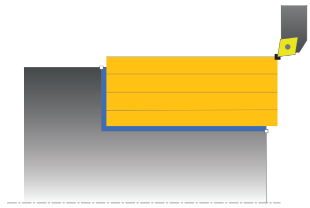
Example

```
11 CYCL DEF 821 SHOULDER, LONGITDNL. ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q493=+50               ;DIAMETER AT CONTOUR END ~
    Q494=-55               ;CONTOUR END IN Z ~
    Q463=+3                ;MAX. CUTTING DEPTH ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q506=+0                ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 R0 FMAX M303
13 CYCL CALL
```

14.8 Cycle 812 SHOULDER, LONG. EXT.

ISO programming
G812

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute longitudinal turning of shoulders.
Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and starts the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand.

1 The control performs a paraxial infeed movement at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

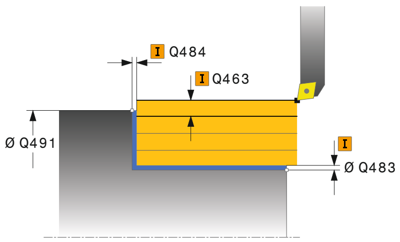
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

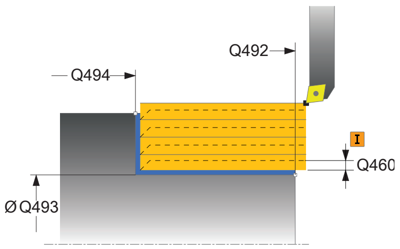
  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

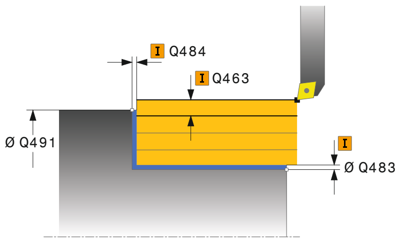
- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters





Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
 	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

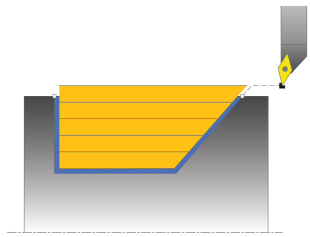
Example

```
11 CYCL DEF 812 SHOULDER, LONG. EXT. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=+0              ;CONTOUR START IN Z ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-55             ;CONTOUR END IN Z ~
    Q495=+5              ;ANGLE OF CIRCUM. SURFACE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+0              ;ANGLE OF FACE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.9 Cycle 813 TURN PLUNGE CONTOUR LONGITUDINAL

ISO programming
G813

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute longitudinal turning of shoulders with plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

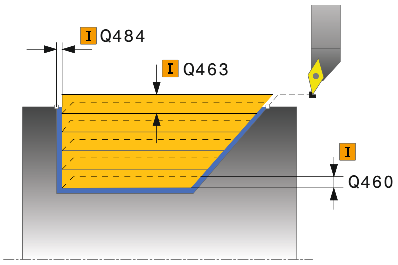
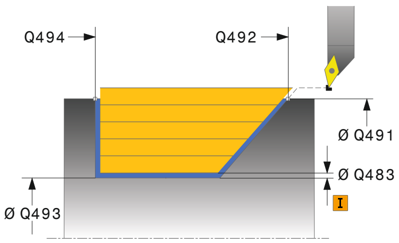
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
  	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is the line perpendicular to the rotary axis. Input: **0...89.9999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**

Help graphic	Parameter
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

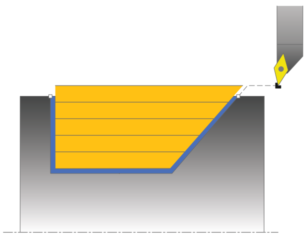
Example

```
11 CYCL DEF 813 TURN PLUNGE CONTOUR LONGITUDINAL ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=-10             ;CONTOUR START IN Z ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-55             ;CONTOUR END IN Z ~
    Q495=+70             ;ANGLE OF SIDE ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 R0 FMAX M303
13 CYCL CALL
```

14.10 Cycle 814 TURN PLUNGE LONGITUDINAL EXT.

ISO programming
G814

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute longitudinal turning of shoulders with plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic | **Parameter**

Q215 Machining operation (0/1/2/3)?

Define extent of machining:

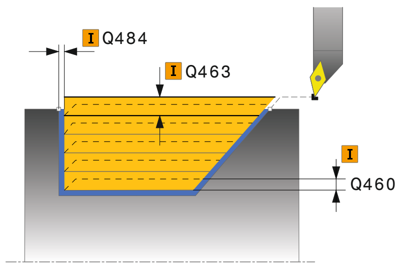
0: Roughing and finishing

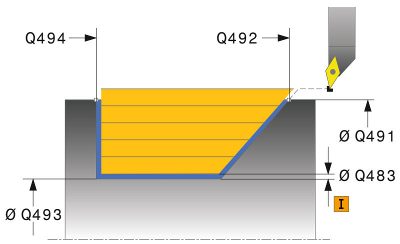
1: Only roughing

2: Only finishing to final dimension

3: Only finishing to oversize

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





Q460 Set-up clearance?

Distance for retraction and prepositioning. This value has an incremental effect.

Input: **0...999.999**

Q491 Diameter at contour start?

X coordinate of the contour starting point (diameter value)

Input: **-99999.999...+99999.999**

Q492 Contour start in Z?

Z coordinate of the starting point for the plunging path

Input: **-99999.999...+99999.999**

Q493 Diameter at end of contour?

X coordinate of the contour end point (diameter value)

Input: **-99999.999...+99999.999**

Q494 Contour end in Z?

Z coordinate of the contour end point

Input: **-99999.999...+99999.999**

Q495 Angle of side?

Angle of plunging flank. The reference angle is the line perpendicular to the rotary axis.

Input: **0...89.9999**

Q501 Starting element type (0/1/2)?

Define the type of element at the beginning of the contour (circumferential surface):

0: No additional element

1: Element is a chamfer

2: Element is a radius

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

Q502 Size of starting element?

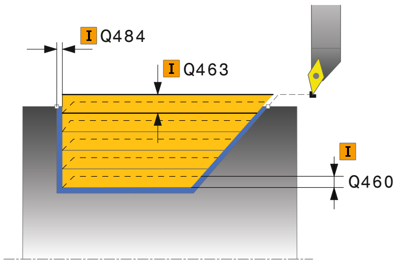
Size of the starting element (chamfer section)

Input: **0...999.999**

Q500 Radius of the contour corner?

Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert.

Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

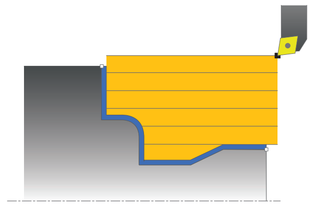
Example

```
11 CYCL DEF 814 TURN PLUNGE LONGITUDINAL EXT. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=-10             ;CONTOUR START IN Z ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-55             ;CONTOUR END IN Z ~
    Q495=+70             ;ANGLE OF SIDE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+0              ;ANGLE OF FACE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.11 Cycle 810 TURN CONTOUR LONG.

ISO programming
G810

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute longitudinal turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

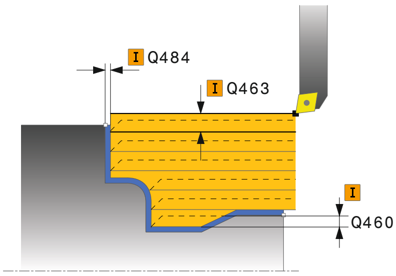
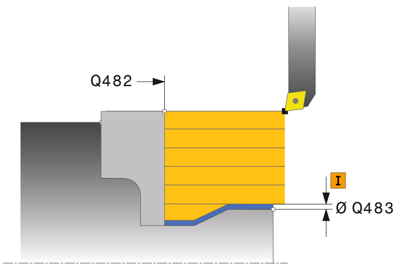
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

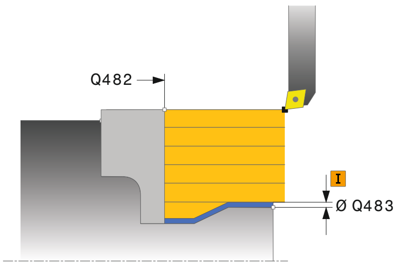
  Further information: "Fundamentals of turning cycles", Page 563

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q487 Allow plunging (0/1)? Permit the machining of plunging elements: **0**: Do not machine any plunging elements **1**: Machine plunging elements Input: **0**, **1**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480**/**Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	**Q482 Value of cutting limit in Z?** Z value for contour limit Input: **-99999.999...+99999.999**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

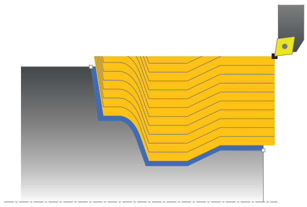
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 810 TURN CONTOUR LONG. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q487=+1                 ;PLUNGE ~
    Q488=+0                 ;PLUNGING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q506=+0                 ;CONTOUR SMOOTHING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Z-35
22 RND R5
23 L  X+50  Z-40
24 L  Z-55
25 CC  X+60  Z-55
26 C  X+60  Z-60
27 L  X+100
28 LBL 0
```

14.12 Cycle 815 CONTOUR-PAR. TURNING

ISO programming
G815

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning with roughing is contour-parallel.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and end point. The cut is performed in contour-parallel mode at the defined feed rate **Q478**.
3 The control returns the tool at the defined feed rate back to the starting position in the X coordinate.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter

Q215 Machining operation (0/1/2/3)?

Define extent of machining:

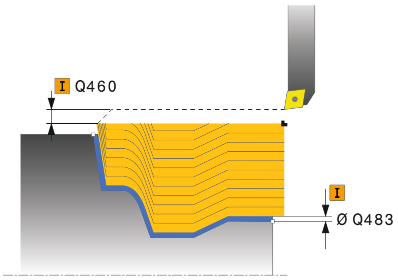
0: Roughing and finishing

1: Only roughing

2: Only finishing to final dimension

3: Only finishing to oversize

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



Q460 Set-up clearance?

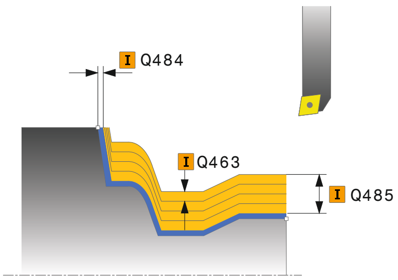
Distance for retraction and prepositioning. This value has an incremental effect.

Input: **0...999.999**

Q485 Allowance for workpiece blank?

Contour-parallel oversize on the defined contour. This value has an incremental effect.

Input: **0...99.999**



Q486 Type of cut lines (=0/1)?

Define the type of cutting lines:

0: Cuts with consistent chip cross section

1: Equidistance cut distribution

Input: **0**, **1**

Q499 Reverse the contour (0-2)?

Define the machining direction of the contour:

0: Contour is executed in the programmed direction

1: Contour is executed in the direction opposite to the programmed direction

2: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted

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

Q463 Maximum cutting depth?

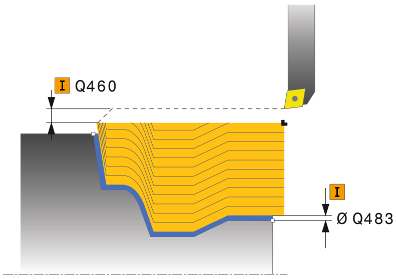
Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts.

Input: **0...99.999**

Q478 Roughing feed rate?

Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.

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

Help graphic	Parameter
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

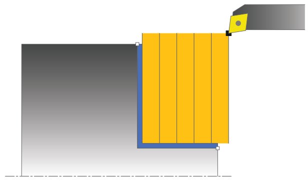
Example

```
11 CYCL DEF 815 CONTOUR-PAR. TURNING ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q485=+5                ;ALLOWANCE ON BLANK ~
    Q486=+0                ;INTERSECTING LINES ~
    Q499=+0                ;REVERSE CONTOUR ~
    Q463=+3                ;MAX. CUTTING DEPTH ~
    Q478=0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.13 Cycle 821 SHOULDER, FACE

ISO programming
G821

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to face turn right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. The control performs a paraxial infeed movement at rapid traverse.
3. The control finishes the contour of the finished part at the defined feed rate **Q505**.
4. The control retracts the tool at the defined feed rate to the set-up clearance.
5. The control returns the tool at rapid traverse to the cycle starting point.

Notes

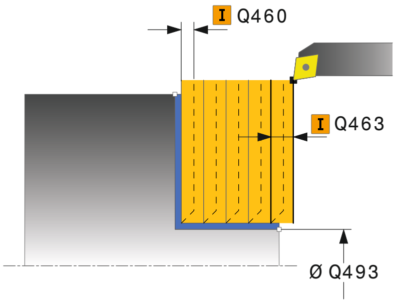
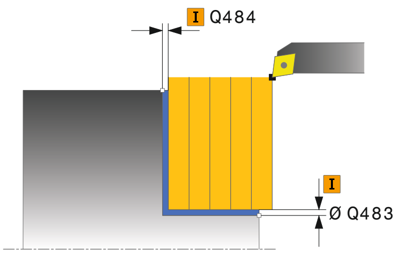
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

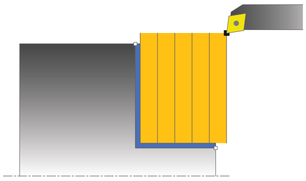
Example

```
11 CYCL DEF 821 SHOULDER, FACE ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q493=+30             ;DIAMETER AT CONTOUR END ~
    Q494=-5              ;CONTOUR END IN Z ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.14 Cycle 822 SHOULDER, FACE. EXT.

ISO programming
G822

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to face turn shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control performs a paraxial infeed movement at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

Notes

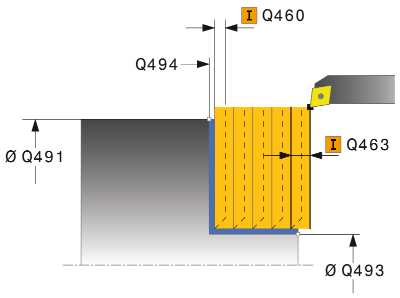
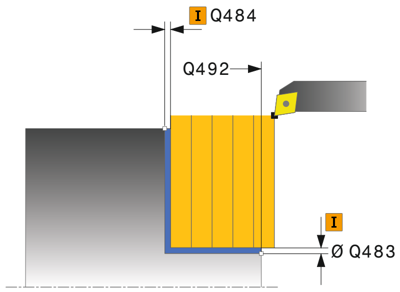
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

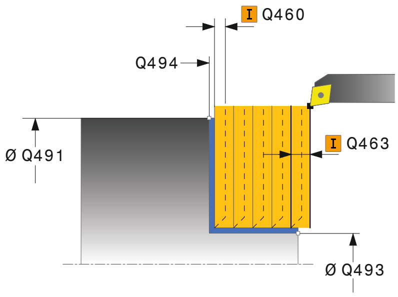
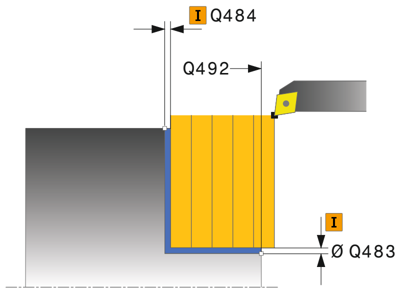
  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	**Q492 Contour start in Z?** Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of the face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

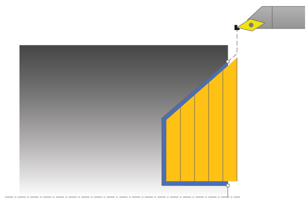
Example

```
11 CYCL DEF 822 SHOULDER, FACE. EXT. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=+0              ;CONTOUR START IN Z ~
    Q493=+30             ;DIAMETER AT CONTOUR END ~
    Q494=-15             ;CONTOUR END IN Z ~
    Q495=+0              ;ANGLE OF FACE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+5              ;ANGLE OF CIRCUM. SURFACE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.15 Cycle 823 TURN TRANSVERSE PLUNGE

ISO programming
G823

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute face turning of plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

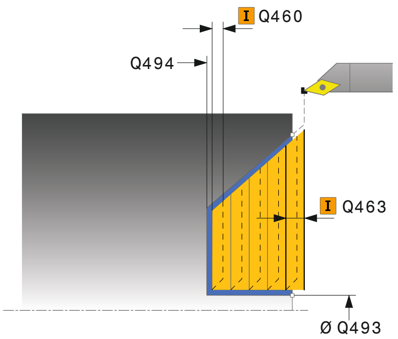
  Further information: "Fundamentals of turning cycles", Page 563

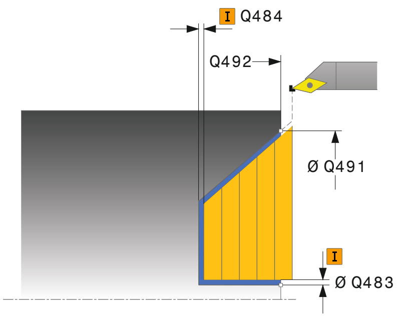
Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter





Q215 Machining operation (0/1/2/3)?
Define extent of machining:
0: Roughing and finishing
1: Only roughing
2: Only finishing to final dimension
3: Only finishing to oversize
Input: **0**, **1**, **2**, **3**

Q460 Set-up clearance?
Distance for retraction and prepositioning. This value has an incremental effect.
Input: **0...999.999**

Q491 Diameter at contour start?
X coordinate of the contour starting point (diameter value)
Input: **-99999.999...+99999.999**

Q492 Contour start in Z?
Z coordinate of the starting point for the plunging path
Input: **-99999.999...+99999.999**

Q493 Diameter at end of contour?
X coordinate of the contour end point (diameter value)
Input: **-99999.999...+99999.999**

Q494 Contour end in Z?
Z coordinate of the contour end point
Input: **-99999.999...+99999.999**

Q495 Angle of side?
Angle of plunging flank. The reference angle is a line parallel to the rotary axis.
Input: **0...89.9999**

Q463 Maximum cutting depth?
Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts.
Input: **0...99.999**

Q478 Roughing feed rate?
Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.
Input: **0...99999.999** or **FAUTO**

Q483 Oversize for diameter?
Diameter oversize on the defined contour. This value has an incremental effect.
Input: **0...99.999**

Help graphic	Parameter
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

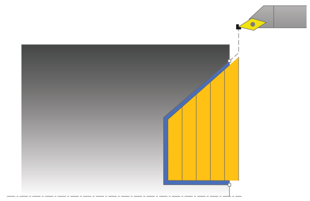
Example

```
11 CYCL DEF 823 TURN TRANSVERSE PLUNGE ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=+0              ;CONTOUR START IN Z ~
    Q493=+20             ;DIAMETER AT CONTOUR END ~
    Q494=-5              ;CONTOUR END IN Z ~
    Q495=+60             ;ANGLE OF SIDE ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.16 Cycle 824 TURN PLUNGE TRANSVERSE EXT.

ISO programming
G824

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute face turning of plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate.
3. The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic | **Parameter**

Q215 Machining operation (0/1/2/3)?

Define extent of machining:

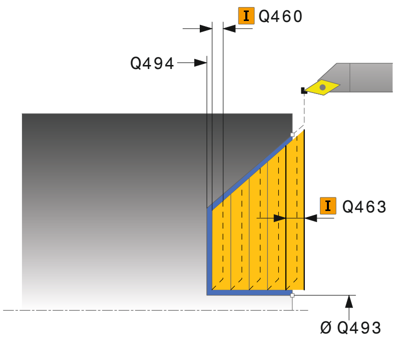
0: Roughing and finishing

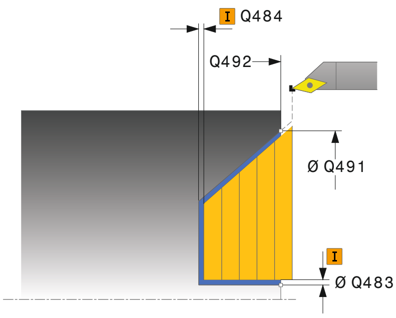
1: Only roughing

2: Only finishing to final dimension

3: Only finishing to oversize

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





Q460 Set-up clearance?

Distance for retraction and prepositioning. This value has an incremental effect.

Input: **0...999.999**

Q491 Diameter at contour start?

X coordinate of the starting point for the plunging path (diameter value)

Input: **-99999.999...+99999.999**

Q492 Contour start in Z?

Z coordinate of the starting point for the plunging path

Input: **-99999.999...+99999.999**

Q493 Diameter at end of contour?

X coordinate of the contour end point (diameter value)

Input: **-99999.999...+99999.999**

Q494 Contour end in Z?

Z coordinate of the contour end point

Input: **-99999.999...+99999.999**

Q495 Angle of side?

Angle of plunging flank. The reference angle is a line parallel to the rotary axis.

Input: **0...89.9999**

Q501 Starting element type (0/1/2)?

Define the type of element at the beginning of the contour (circumferential surface):

0: No additional element

1: Element is a chamfer

2: Element is a radius

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

Q502 Size of starting element?

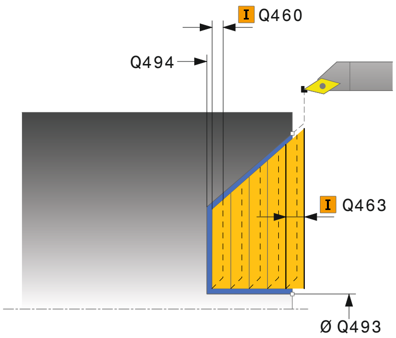
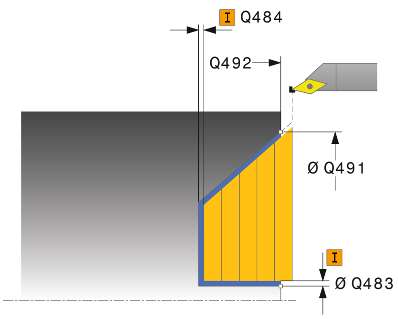
Size of the starting element (chamfer section)

Input: **0...999.999**

Q500 Radius of the contour corner?

Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert.

Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

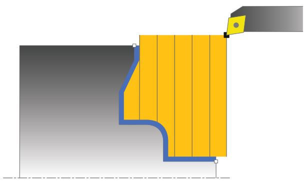
Example

```
11 CYCL DEF 824 TURN PLUNGE TRANSVERSE EXT. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=+0              ;CONTOUR START IN Z ~
    Q493=+20             ;DIAMETER AT CONTOUR END ~
    Q494=-10             ;CONTOUR END IN Z ~
    Q495=+70             ;ANGLE OF SIDE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+0              ;ANGLE OF FACE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.17 Cycle 820 TURN CONTOUR TRANSV.

ISO programming
G820

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute face turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction. The transverse cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

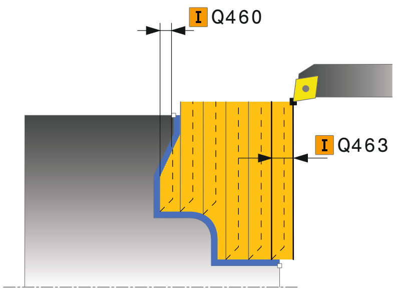
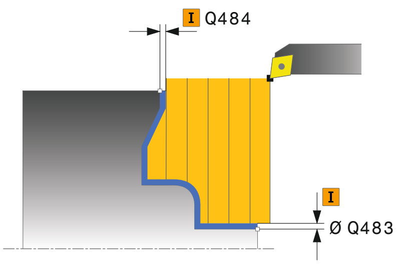
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.

  Further information: "Fundamentals of turning cycles", Page 563

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q487 Allow plunging (0/1)? Permit the machining of plunging elements: **0**: Do not machine any plunging elements **1**: Machine plunging elements Input: **0**, **1**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

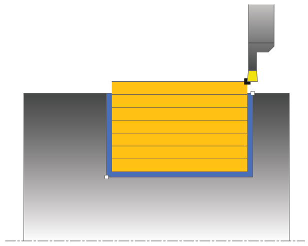
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 820 TURN CONTOUR TRANSV. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q487=+1                 ;PLUNGE ~
    Q488=+0                 ;PLUNGING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q506=+0                 ;CONTOUR SMOOTHING
14 L  X+75  Y+0  Z+2 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+75  Z-20
19 L  X+50
20 RND R2
21 L  X+20  Z-25
22 RND R2
23 L  Z+0
24 LBL 0
```

14.18 Cycle 841 SIMPLE REC. TURNG., RADIAL DIR.

ISO programming
G841

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

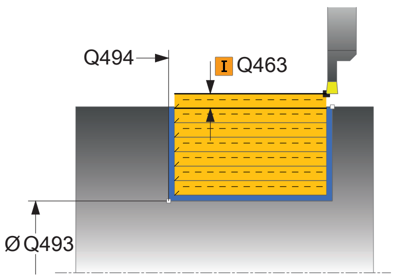
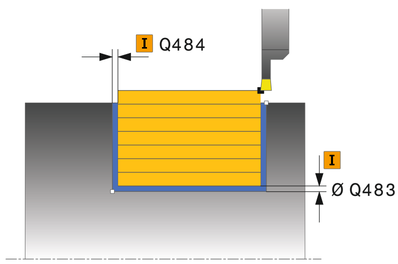
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**

Help graphic	Parameter
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

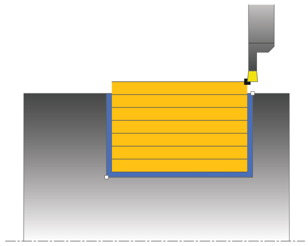
```
11 CYCL DEF 841 SIMPLE REC. TURNG., RADIAL DIR. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-50             ;CONTOUR END IN Z ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q463=+2              ;MAX. CUTTING DEPTH ~
    Q507=+0              ;MACHINING DIRECTION ~
    Q508=+0              ;OFFSET WIDTH ~
    Q509=+0              ;DEPTH COMPENSATION ~
    Q488=+0              ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.19 Cycle 842 ENH.REC.TURNNG, RAD.

ISO programming
G842

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 Diameter at contour start**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Cycle run

Finishing

The control uses the position of the tool at the cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 DIAMETER AT CONTOUR START**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

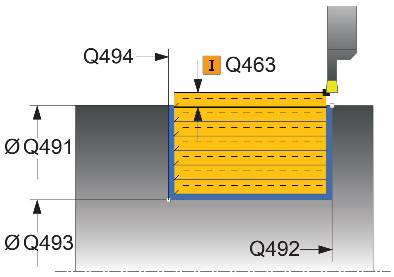
Notes

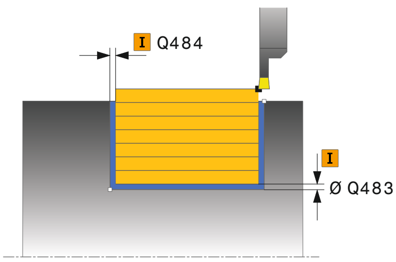
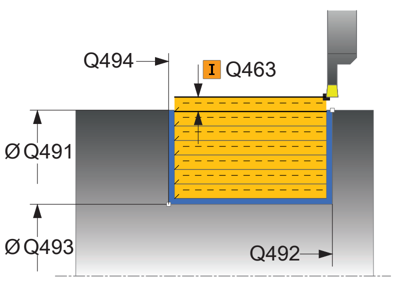
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
 	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and the normal line to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge at the contour end point and the normal line to the rotary axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**

Help graphic	Parameter
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

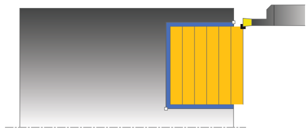
Example

```
11 CYCL DEF 842 EXPND. RECESS, RADL. ~
    Q215=+0            ;MACHINING OPERATION ~
    Q460=+2            ;SAFETY CLEARANCE ~
    Q491=+75           ;DIAMETER AT CONTOUR START ~
    Q492=-20           ;CONTOUR START IN Z ~
    Q493=+50           ;DIAMETER AT CONTOUR END ~
    Q494=-50           ;CONTOUR END IN Z ~
    Q495=+5            ;ANGLE OF SIDE ~
    Q501=+1            ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5          ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5          ;RADIUS OF CONTOUR EDGE ~
    Q496=+5            ;ANGLE OF SECOND SIDE ~
    Q503=+1            ;TYPE OF END ELEMENT ~
    Q504=+0.5          ;SIZE OF END ELEMENT ~
    Q478=+0.3          ;ROUGHING FEED RATE ~
    Q483=+0.4          ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2          ;OVERSIZE IN Z ~
    Q505=+0.2          ;FINISHING FEED RATE ~
    Q463=+2            ;MAX. CUTTING DEPTH ~
    Q507=+0            ;MACHINING DIRECTION ~
    Q508=+0            ;OFFSET WIDTH ~
    Q509=+0            ;DEPTH COMPENSATION ~
    Q488=+0            ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.20 Cycle 851 SIMPLE REC TURNG, AX

ISO programming
G851

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to recess right-angled slots in transverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

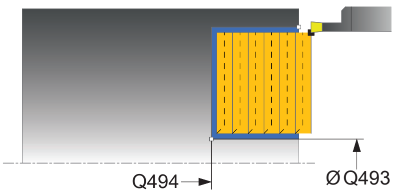
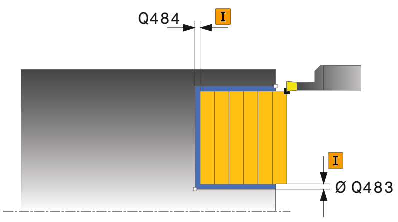
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
 	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
 	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**

Help graphic	Parameter
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

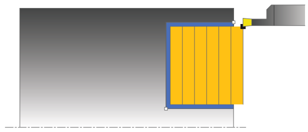
Example

```
11 CYCL DEF 851 SIMPLE REC TURNG, AX ~
    Q215=+0            ;MACHINING OPERATION ~
    Q460=+2            ;SAFETY CLEARANCE ~
    Q493=+50           ;DIAMETER AT CONTOUR END ~
    Q494=-10           ;CONTOUR END IN Z ~
    Q478=+0.3          ;ROUGHING FEED RATE ~
    Q483=+0.4          ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2          ;OVERSIZE IN Z ~
    Q505=+0.2          ;FINISHING FEED RATE ~
    Q463=+2            ;MAX. CUTTING DEPTH ~
    Q507=+0            ;MACHINING DIRECTION ~
    Q508=+0            ;OFFSET WIDTH ~
    Q509=+0            ;DEPTH COMPENSATION ~
    Q488=+0            ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.21 Cycle 852 ENH.REC.TURNING, AX.

ISO programming
G852

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to recess right-angled slots in transverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed. The machining process thus requires a minimum of retraction and infeed movements. Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

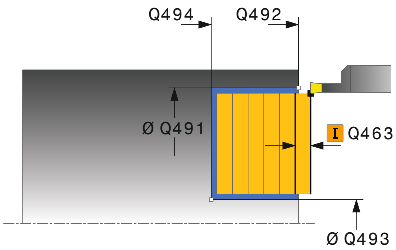
Notes

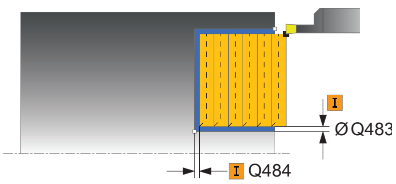
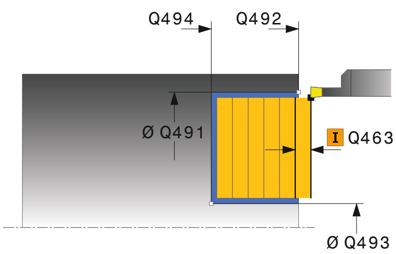
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and a line parallel to the turning axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge of the contour end point and a line parallel to the turning axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**

Help graphic	Parameter
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 852 ENH.REC.TURNING, AX. ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q491=+75               ;DIAMETER AT CONTOUR START ~
    Q492=-20               ;CONTOUR START IN Z ~
    Q493=+50               ;DIAMETER AT CONTOUR END ~
    Q494=-50               ;CONTOUR END IN Z ~
    Q495=+5                ;ANGLE OF SIDE ~
    Q501=+1                ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5              ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5              ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                ;ANGLE OF SECOND SIDE ~
    Q503=+1                ;TYPE OF END ELEMENT ~
    Q504=+0.5              ;SIZE OF END ELEMENT ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q463=+2                ;MAX. CUTTING DEPTH ~
    Q507=+0                ;MACHINING DIRECTION ~
    Q508=+0                ;OFFSET WIDTH ~
    Q509=+0                ;DEPTH COMPENSATION ~
    Q488=+0                ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.22 Cycle 840 RECESS TURNG, RADIAL

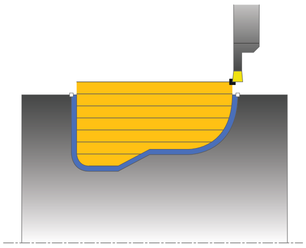
ISO programming
G840

Application

This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the X coordinate of the starting point is less than the contour starting point, the control positions the tool in the X coordinate to the contour starting point and begins the cycle there.

1 The control positions the tool at rapid traverse in the Z coordinate (first recessing position).
2 The control performs a recessing traverse until the first plunging depth is reached.
3 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
4 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6 The tool recesses to the next plunging depth.
7 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

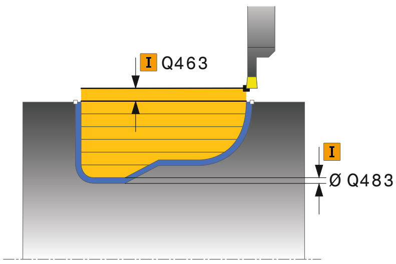
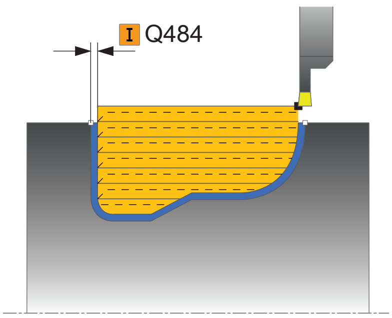
▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	**Q484 Oversize in Z?** Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**

Help graphic	Parameter
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q499 Reverse contour (0=no/1=yes)? Machining direction: **0**: Machining in the direction of contour **1**: Machining in the direction opposite to the contour direction Input: **0**, **1**

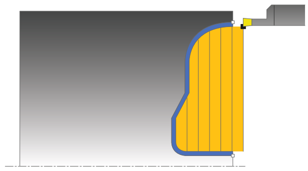
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 840 RECESS TURNG, RADIAL ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q488=+0                ;PLUNGING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q479=+0                ;CONTOUR MACHINING LIMIT ~
    Q480=+0                ;DIAMETER LIMIT VALUE ~
    Q482=+0                ;LIMIT VALUE Z ~
    Q463=+2                ;MAX. CUTTING DEPTH ~
    Q507=+0                ;MACHINING DIRECTION ~
    Q508=+0                ;OFFSET WIDTH ~
    Q509=+0                ;DEPTH COMPENSATION ~
    Q499=+0                ;REVERSE CONTOUR
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z-10
19 L  X+40  Z-15
20 RND R3
21 CR  X+40  Z-35 R+30 DR+
22 RND R3
23 L  X+60  Z-40
24 LBL 0
```

14.23 Cycle 850 RECESS TURNG, AXIAL

ISO programming
G850

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to machine slots of any shape in transverse direction by recess turning. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1. The control positions the tool at rapid traverse in the X coordinate (first recessing position).
2. The control performs a recessing traverse until the first plunging depth is reached.
3. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
4. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6. The tool recesses to the next plunging depth.
7. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side walls of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate.
4. The control returns the tool at rapid traverse to the cycle starting point.

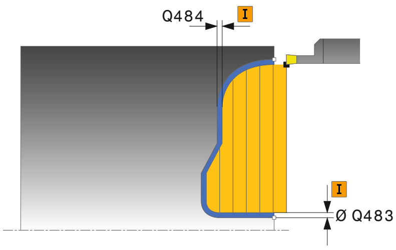
Notes

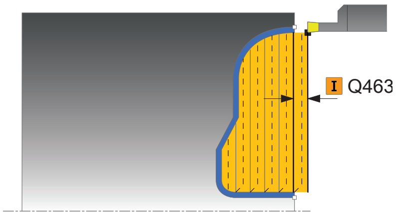
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**

Help graphic	Parameter
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q499 Reverse contour (0=no/1=yes)? Machining direction: **0**: Machining in the direction of contour **1**: Machining in the direction opposite to the contour direction Input: **0**, **1**

Example

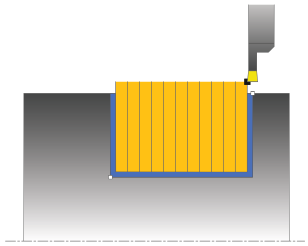
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 850 RECESS TURNG, AXIAL ~
    Q215=+0             ;MACHINING OPERATION ~
    Q460=+2             ;SAFETY CLEARANCE ~
    Q478=+0.3           ;ROUGHING FEED RATE ~
    Q488=0              ;PLUNGING FEED RATE ~
    Q483=+0.4           ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2           ;OVERSIZE IN Z ~
    Q505=+0.2           ;FINISHING FEED RATE ~
    Q479=+0             ;CONTOUR MACHINING LIMIT ~
    Q480=+0             ;DIAMETER LIMIT VALUE ~
    Q482=+0             ;LIMIT VALUE Z ~
    Q463=+2             ;MAX. CUTTING DEPTH ~
    Q507=+0             ;MACHINING DIRECTION ~
    Q508=+0             ;OFFSET WIDTH ~
    Q509=+0             ;DEPTH COMPENSATION ~
    Q499=+0             ;REVERSE CONTOUR
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Y-15
22 L  Z+0
23 LBL 0
```

14.24 Cycle 861 SIMPLE RECESS, RADL.

ISO programming
G861

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to radially cut in right-angled slots.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

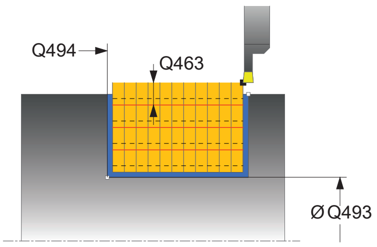
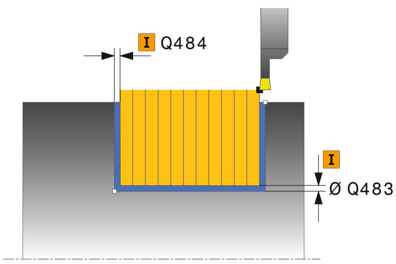
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

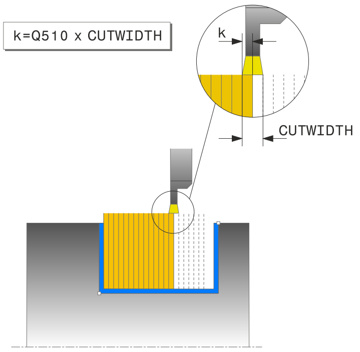
Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**

Help graphic	Parameter



Q510 Overlap factor for recess width?

Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k".

Input: **0.001...1**

Q511 Feed rate factor in %?

Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**.

If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times.

Input: **0.001...150**

Q462 Retraction behavior (0/1)?

With **Q462**, you define the retraction behavior after the recess.

0: The control retracts the tool along the contour

1: The control first moves the tool at an angle away from the contour and then retracts it

Input: **0**, **1**

Q211 Dwell time / 1/min?

A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is performed only after the tool has remained for **Q211** revolutions.

Input: **0...999.99**

Q562 Multiple plunging (0/1)?

0: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**)

1: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment

Input: **0**, **1**

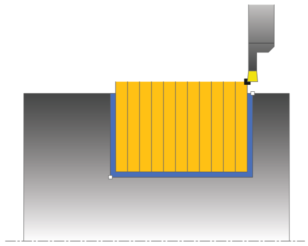
Example

```
11 CYCL DEF 861 SIMPLE RECESS, RADL. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-50             ;CONTOUR END IN Z ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q463=+0              ;LIMIT TO DEPTH ~
    Q510=+0.8            ;RECESSING OVERLAP ~
    Q511=+100            ;FEED RATE FACTOR ~
    Q462=0               ;RETRACTION MODE ~
    Q211=3               ;DWELL TIME IN REVS ~
    Q562=+0              ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.25 Cycle 862 EXPND. RECESS, RADL.

ISO programming
G862

Application



Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to radially cut in slots. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

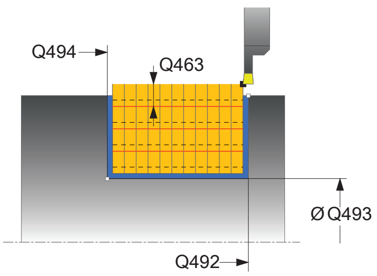
Notes

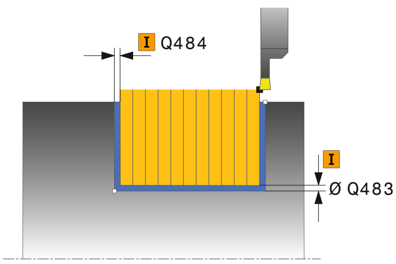
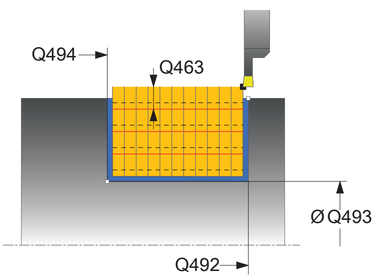
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

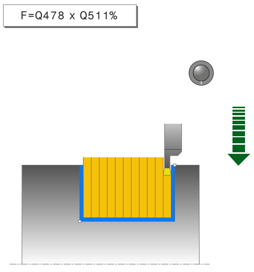
Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	**Q215 Machining operation (0/1/2/3)?** Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and the normal line to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge at the contour end point and the normal line to the rotary axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	**Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is performed only after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

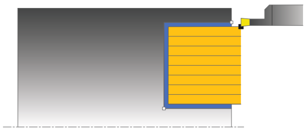
Example

```
11 CYCL DEF 862 EXPND. RECESS, RADL. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=-20             ;CONTOUR START IN Z ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-50             ;CONTOUR END IN Z ~
    Q495=+5              ;ANGLE OF SIDE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+5              ;ANGLE OF SECOND SIDE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q463=+0              ;LIMIT TO DEPTH ~
    Q510=0.8             ;RECESSING OVERLAP ~
    Q511=+100            ;FEED RATE FACTOR ~
    Q462=+0              ;RETRACTION MODE ~
    Q211=3               ;DWELL TIME IN REVS ~
    Q562=+0              ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.26 Cycle 871 SIMPLE RECESS, AXIAL

ISO programming
G871

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to perform axial recessing of right-angled slots (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

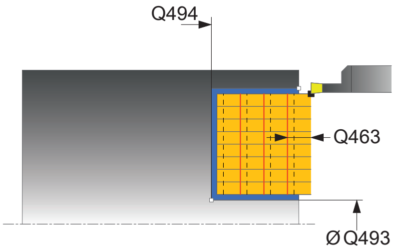
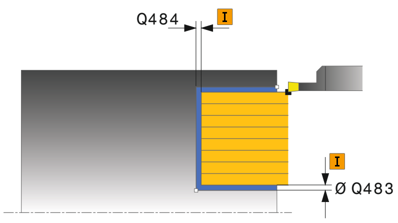
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is performed only after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

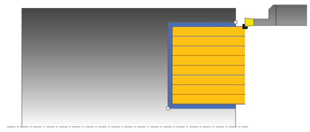
Example

```
11 CYCL DEF 871 SIMPLE RECESS, AXIAL ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-10             ;CONTOUR END IN Z ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q463=+0              ;LIMIT TO DEPTH ~
    Q510=+0,8            ;RECESSING OVERLAP ~
    Q511=+100            ;FEED RATE FACTOR ~
    Q462=0               ;RETRACTION MODE ~
    Q211=3               ;DWELL TIME IN REVS ~
    Q562=+0              ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.27 Cycle 872 EXPND. RECESS, AXIAL

ISO programming
G872

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to perform axial recessing of slots (face recessing). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control retracts the tool at rapid traverse.
4 The control positions the tool at rapid traverse to the second slot side.
5 The control finishes the side wall of the slot at the defined feed rate **Q505**.
6 The control finishes one half of the slot at the defined feed rate.
7 The control positions the tool at rapid traverse to the first side.
8 The control finishes the other half of the slot at the defined feed rate.
9 The control returns the tool at rapid traverse to the cycle starting point.

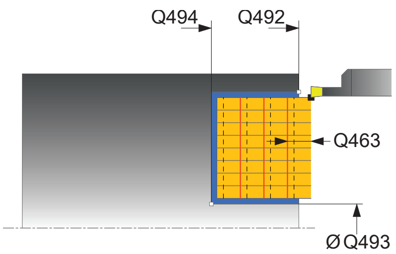
Notes

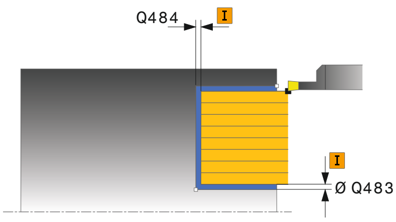
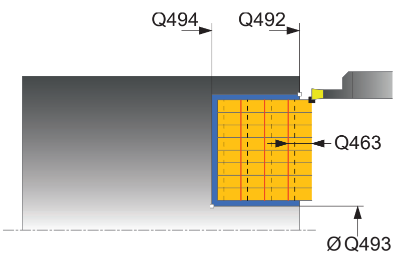
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
 	**Q492 Contour start in Z?** Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and a line parallel to the turning axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge of the contour end point and a line parallel to the turning axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is performed only after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

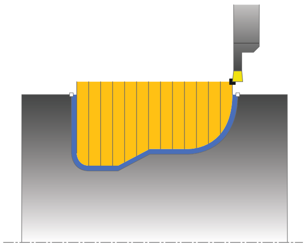
```
11 CYCL DEF 872 EXPND. RECESS, AXIAL ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=-20             ;CONTOUR START IN Z ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-50             ;CONTOUR END IN Z ~
    Q495=+5              ;ANGLE OF SIDE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+5              ;ANGLE OF SECOND SIDE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q463=+0              ;LIMIT TO DEPTH ~
    Q510=+0.08           ;RECESSING OVERLAP ~
    Q511=+100            ;FEED RATE FACTOR ~
    Q462=+0              ;RETRACTION MODE ~
    Q211=+3              ;DWELL TIME IN REVS ~
    Q562=+0              ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.28 Cycle 860 CONT. RECESS, RADIAL

ISO programming
G860

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to radially cut in slots of any form.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

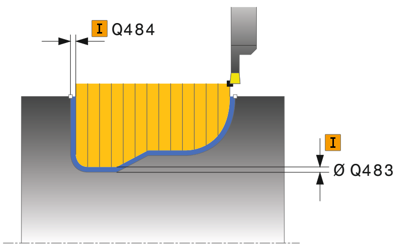
▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

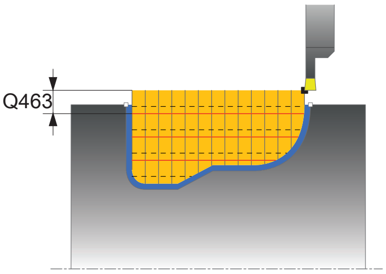
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**

Help graphic	Parameter
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is performed only after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

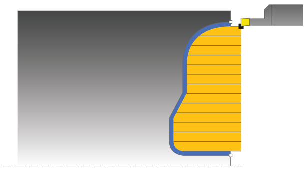
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 860 CONT. RECESS, RADIAL ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q479=+0                ;CONTOUR MACHINING LIMIT ~
    Q480=+0                ;DIAMETER LIMIT VALUE ~
    Q482=+0                ;LIMIT VALUE Z ~
    Q463=+0                ;LIMIT TO DEPTH ~
    Q510=0.08              ;RECESSING OVERLAP ~
    Q511=+100              ;FEED RATE FACTOR ~
    Q462=+0                ;RETRACTION MODE ~
    Q211=3                 ;DWELL TIME IN REVS ~
    Q562=+0                ;MULTIPLE PLUNGING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z-20
19 L  X+45
20 RND R2
21 L  X+40  Y-25
22 L  Z+0
23 LBL 0
```

14.29 Cycle 870 CONT. RECESS, AXIAL

ISO programming
G870

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to perform axial recessing of slots of any form (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes one half of the slot at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes the other half of the slot at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

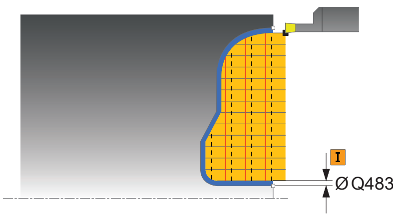
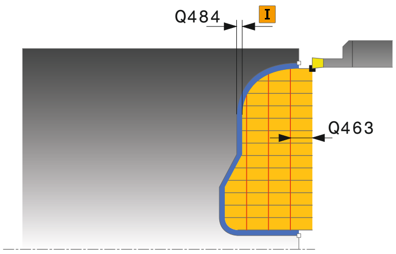
- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.
- Finishing the contour requires programming tool radius compensation **RL** or **RR** in the contour description.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	**Q484 Oversize in Z?** Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**

Help graphic	Parameter
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is performed only after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

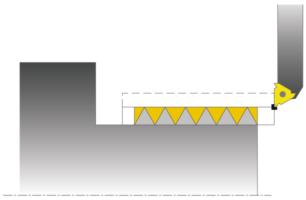
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 870 CONT. RECESS, AXIAL ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q463=+0                 ;LIMIT TO DEPTH ~
    Q510=+0.8               ;RECESSING OVERLAP ~
    Q511=+100               ;FEED RATE FACTOR ~
    Q462=+0                 ;RETRACTION MODE ~
    Q211=+3                 ;DWELL TIME IN REVS ~
    Q562=+0                 ;MULTIPLE PLUNGING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Y-15
22 L  Z+0
23 LBL 0
```

14.30 Cycle 831 THREAD LONGITUDINAL

ISO programming
G831

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to execute longitudinal turning of threads.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth, the cycle uses thread depth in accordance with the ISO1502 standard.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a paraxial longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

While the control cuts a thread, the feed-rate override knob is disabled. The spindle-speed override knob is still active to a limited extent.

Notes

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180° (for example, to machine internal and external threads with only one tool). If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position. There is a danger of collision!

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Caution: Danger to the tool and workpiece!

If you program an angle of infeed **Q467** wider than the side angle of the thread, this may destroy the thread flanks. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

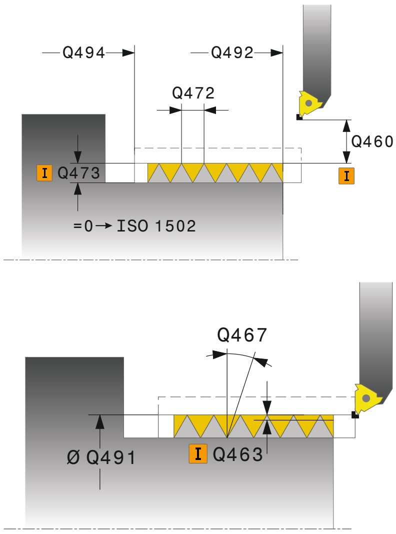
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The number of threads for thread cutting is limited to 500.
- In Cycle **832 THREAD EXTENDED**, parameters are available for approach and overrun.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The control uses the set-up clearance **Q460** as approach length. The approach path must be long enough for the feed axes to be accelerated to the required velocity.
- The control uses the thread pitch as idle travel path. The idle travel distance must be long enough to decelerate the feed axes.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.

Cycle parameters

Help graphic	Parameter
	**Q471 Thread position (0=ext./1=int.)?** Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q460 Setup clearance? Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) until the synchronized feed rate is reached. Input: **0...999.999**
	Q491 Thread diameter? Define the nominal diameter of the thread. Input: **0.001...99999.999**
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	Q492 Contour start in Z? Z coordinate of the starting point Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the end point, including the thread runout **Q474** Input: **-99999.999...+99999.999**
	Q474 Length of thread runout? Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q463 Maximum cutting depth? Maximum plunging depth in radial direction relative to the radius. Input: **0.001...999.999**
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is the line perpendicular to the rotary axis. Input: **0...60**

Help graphic	Parameter
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

Example

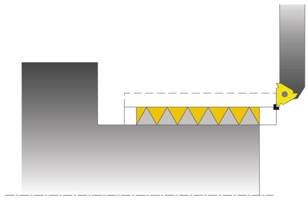
```
11 CYCL DEF 831 THREAD LONGITUDINAL ~
    Q471=+0             ;THREAD POSITION ~
    Q460=+5             ;SAFETY CLEARANCE ~
    Q491=+75            ;THREAD DIAMETER ~
    Q472=+2             ;THREAD PITCH ~
    Q473=+0             ;DEPTH OF THREAD ~
    Q492=+0             ;CONTOUR START IN Z ~
    Q494=-15            ;CONTOUR END IN Z ~
    Q474=+0             ;THREAD RUN-OUT ~
    Q463=+0.5           ;MAX. CUTTING DEPTH ~
    Q467=+30            ;ANGLE OF INFEED ~
    Q468=+0             ;TYPE OF INFEED ~
    Q470=+0             ;STARTING ANGLE ~
    Q475=+30            ;NUMBER OF STARTS ~
    Q476=+30            ;NUMBER OF AIR CUTS
12 L  X+80  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.31 Cycle 832 THREAD EXTENDED

ISO programming
G832

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to execute both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- Selection of a longitudinal thread or transversal thread
- The parameters for dimension type of taper, taper angle, and contour starting point X enable the definition of various tapered threads
- The parameters for the approach length and the idle travel distance define a path in which feed axes can be accelerated and decelerated

You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1. The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2. The control performs a longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3. The control retracts the tool at rapid traverse to the set-up clearance.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6. The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7. The control performs the number of air cuts as defined in **Q476**.
8. The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9. The control returns the tool at rapid traverse to the cycle starting point.

While the control cuts a thread, the feed-rate override knob is disabled. The spindle-speed override knob is still active to a limited extent.

Notes

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180° (for example, to machine internal and external threads with only one tool). If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position. There is a danger of collision!

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Caution: Danger to the tool and workpiece!

If you program an angle of infeed **Q467** wider than the side angle of the thread, this may destroy the thread flanks. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

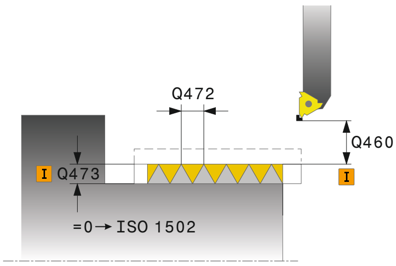
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

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

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (**Q466**) must be long enough to decelerate the feed axes.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.

Cycle parameters

Help graphic	Parameter
	Q471 Thread position (0=ext./1=int.)? Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q461 Thread orientation (0/1)? Define the direction of the thread pitch: **0**: L (parallel to the turning axis) **1**: Perpendicular (perpendicular to the turning axis) Input: **0**, **1**
	**Q460 Set-up clearance?** Set-up clearance perpendicular to the thread pitch Input: **0...999.999**
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	Q464 Dimens. type taper (0-4)? Type of dimensioning of the taper contour: **0**: Via start and end point **1**: Via end point, start X and angle of taper **2**: Via end point, start Z and angle of taper **3**: Via start point, end X and angle of taper **4**: Via start point, end Z and angle of taper Input: **0**, **1**, **2**, **3**, **4**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the end point Input: **-99999.999...+99999.999**

Help graphic	Parameter
	Q469 Taper angle (diameter)? Taper angle of the contour Input: **-180...+180**
	Q474 Length of thread runout? Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q465 Starting path? Length of the path in the direction of the pitch at which the feed axes are accelerated to the required speed. The approach path is outside of the defined thread contour. This value has an incremental effect. Input: **0.1...99.9**
	Q466 Overrun path? Input: **0.1...99.9**
	Q463 Maximum cutting depth? Maximum infeed perpendicular to the thread pitch Input: **0.001...999.999**
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is formed by the parallel line to the thread pitch. Input: **0...60**
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

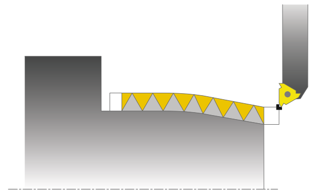
Example

```
11 CYCL DEF 832 THREAD EXTENDED ~
    Q471=+0                 ;THREAD POSITION ~
    Q461=+0                 ;THREAD ORIENTATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q472=+2                 ;THREAD PITCH ~
    Q473=+0                 ;DEPTH OF THREAD ~
    Q464=+0                 ;DIMENSION TYPE TAPER ~
    Q491=+100               ;DIAMETER AT CONTOUR START ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q493=+110               ;DIAMETER AT CONTOUR END ~
    Q494=-35                ;CONTOUR END IN Z ~
    Q469=+0                 ;TAPER ANGLE ~
    Q474=+0                 ;THREAD RUN-OUT ~
    Q465=+4                 ;STARTING PATH ~
    Q466=+4                 ;OVERRUN PATH ~
    Q463=+0.5               ;MAX. CUTTING DEPTH ~
    Q467=+30                ;ANGLE OF INFEED ~
    Q468=+0                 ;TYPE OF INFEED ~
    Q470=+0                 ;STARTING ANGLE ~
    Q475=+30                ;NUMBER OF STARTS ~
    Q476=+30                ;NUMBER OF AIR CUTS
12 L  X+80  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.32 Cycle 830 THREAD CONTOUR-PARALLEL

ISO programming
G830

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

This cycle enables you to execute both face turning and longitudinal turning of threads with any shape.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control runs a thread cut parallel to the defined thread contour. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

> While the control cuts a thread, the feed-rate override knob is disabled. The spindle-speed override knob is still active to a limited extent.

Notes

NOTICE

Danger of collision!

Cycle **830** executes the overrun **Q466** following the programmed contour. There is a danger of collision!

- Clamp the workpiece in such a way that there is no danger of collision if the control extends the contour by **Q466**, **Q467**.

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180° (for example, to machine internal and external threads with only one tool). If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position. There is a danger of collision!

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Caution: Danger to the tool and workpiece!

If you program an angle of infeed **Q467** wider than the side angle of the thread, this may destroy the thread flanks. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

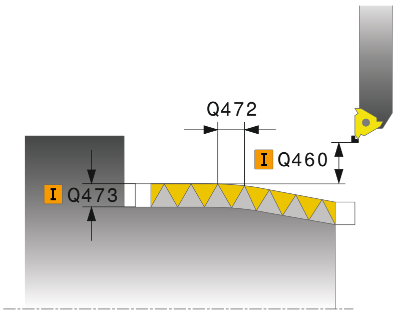
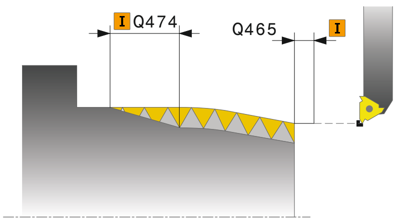
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- Both the approach and overrun take place outside the defined contour.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (**Q466**) must be long enough to decelerate the feed axes.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.
- 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
	Q471 Thread position (0=ext./1=int.)? Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q461 Thread orientation (0/1)? Define the direction of the thread pitch: **0**: L (parallel to the turning axis) **1**: Perpendicular (perpendicular to the turning axis) Input: **0**, **1**
	**Q460 Set-up clearance?** Set-up clearance perpendicular to the thread pitch Input: **0...999.999**
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	**Q474 Length of thread runout?** Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q465 Starting path? Length of the path in the direction of the pitch at which the feed axes are accelerated to the required speed. The approach path is outside of the defined thread contour. This value has an incremental effect. Input: **0.1...99.9**
	Q466 Overrun path? Input: **0.1...99.9**
	Q463 Maximum cutting depth? Maximum infeed perpendicular to the thread pitch Input: **0.001...999.999**

Help graphic	Parameter
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is formed by the parallel line to the thread pitch. Input: **0...60**
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

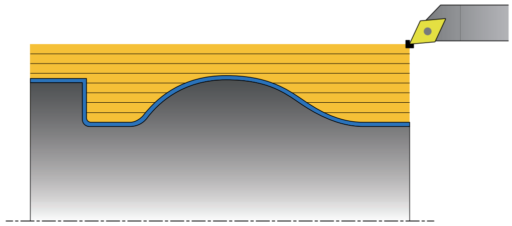
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 830 THREAD CONTOUR-PARALLEL ~
    Q471=+0                ;THREAD POSITION ~
    Q461=+0                ;THREAD ORIENTATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q472=+2                ;THREAD PITCH ~
    Q473=+0                ;DEPTH OF THREAD ~
    Q474=+0                ;THREAD RUN-OUT ~
    Q465=+4                ;STARTING PATH ~
    Q466=+4                ;OVERRUN PATH ~
    Q463=+0.5              ;MAX. CUTTING DEPTH ~
    Q467=+30               ;ANGLE OF INFEED ~
    Q468=+0                ;TYPE OF INFEED ~
    Q470=+0                ;STARTING ANGLE ~
    Q475=+30               ;NUMBER OF STARTS ~
    Q476=+30               ;NUMBER OF AIR CUTS
14 L  X+80  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  X+70  Z-30
20 RND R60
21 L  Z-45
22 LBL 0
```

14.33 Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option 158)

ISO programming
G882

Application



> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

In Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**, the defined contour area is roughed simultaneously in several steps using a movement that includes at least 3 axes (two linear axes and one rotary axis). This allows machining of complex contours with a single tool. During machining, the cycle continuously adjusts the tool angle of inclination based on the following criteria:

- Avoiding collisions between the workpiece, the tool, and the tool carrier
- The tooth does not suffer single-spot wear
- Undercuts are possible

Execution with a FreeTurn tool

You can execute this cycle with FreeTurn tools. This method allows you to perform the most common turning operations with just one tool. Machining times can be reduced through the flexible tool because fewer tool changes occur.

Requirements:

- This function must be adapted by your machine manufacturer.
- You must properly define the tool.

> The NC program remains unchanged except for the calling of the FreeTurn cutting edges, see "Example: Turning with a FreeTurn tool", Page 715

Roughing cycle sequence

1. The cycle positions the tool at the cycle start position (tool position when the cycle is called), taking the first tool angle of inclination into account. Then, the tool moves to set-up clearance. If the angle of inclination cannot be achieved at the cycle start position, the control first moves the tool to set-up clearance and from there tilts it using the first tool angle of inclination.
2. The tool moves to the plunging depth **Q519**. The profile infeed may be exceeded for a short time up to the value of **Q463 MAX. CUTTING DEPTH** (for example, when machining a corner).
3. The contour is roughed simultaneously using the roughing feed-rate in **Q478**. If you define the plunging feed rate **Q488** in the cycle, it will be effective for the plunging elements. Machining depends on the following input parameters:
   - **Q590**: **MACHINING MODE**
   - **Q591**: **MACHINING SEQUENCE**
   - **Q389**: **UNI.- BIDIRECTIONAL**
4. After each infeed, the control lifts the tool in rapid traverse by the set-up clearance value.
5. The control repeats steps 2 to 4 until the contour has been machined completely.
6. The control retracts the tool at the machining feed rate by the set-up clearance value and then moves it with rapid traverse to the starting position (first in the X axis and then in the Z axis direction)

Notes

NOTICE

Danger of collision!

The cycle uses the position of the tool at cycle call as the cycle starting position. Incorrect pre-positioning can cause contour damage. There is a danger of collision!

- Move the tool to a safe position in the X and Z axes.

NOTICE

Danger of collision!

If the contour ends too closely at the fixture, a collision between tool and fixture might occur during machining.

- When clamping, take both the tool angle of inclination and the departure movement into account

NOTICE

Risk of collision!

Collision monitoring only considers the two-dimensional X-Z working plane. The cycle does not check for collisions with an area in the Y coordinate of the cutting edge, tool holder, or tilting body.

- Verify the NC program in **SINGLE BLOCK**
- Limit the machining area

NOTICE

Danger of collision!

Depending on the geometry of the cutting edge, residual material may be left over. Danger of collision during subsequent machining operations!

- Run a simulation to verify the sequence and the contour

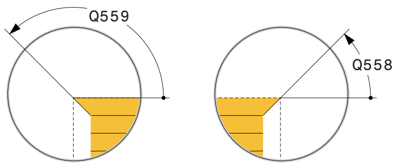
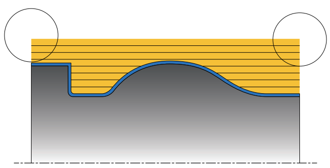
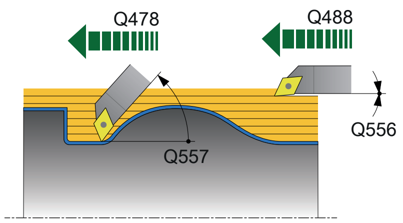
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- If you programmed **M136** before the cycle call, the control interprets the feed rate in millimeters per revolution.
- Software limit switches limit the possible inclination angles **Q556** and **Q557**. If, in **Test Run** the software limit switches are deactivated, then the simulation may deviate from the later machining operation.
- If it is not possible to machine a particular contour area using this cycle, the control tries to divide the contour area into subareas that can be reached so as to machine them individually.

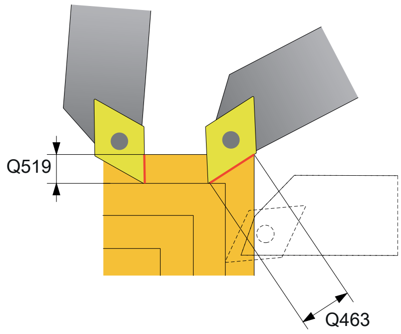
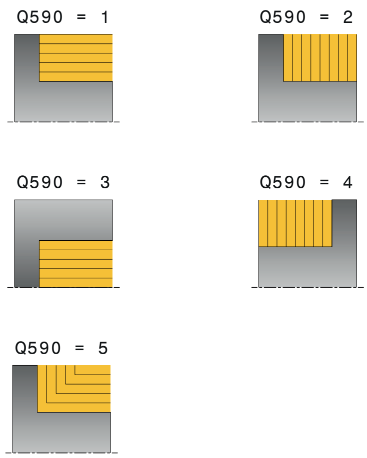
Notes on programming

- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- Prior to the cycle call, you must program **FUNCTION TCPM**. HEIDENHAIN recommends programming the tool reference point **REFPNT TIP-CENTER** in **FUNCTION TCPM**. Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip.
  Further information: User's Manual for **Klartext Programming**
- The cycle requires a radius compensation (**RL**/**RR**) in its contour description.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- For determining the inclination angle, the cycle requires the definition of a tool holder. For this purpose, assign a tool holder to the tool in the **KINEMATIC** column of the tool table.

  Further information: User's Manual for **Setup, Testing and Running NC Programs**
- Define a value in **Q463 MAX. CUTTING DEPTH** relative to the cutting edge because, depending on the tool inclination, the infeed from **Q519** may be temporarily exceeded. Use this parameter to limit the extent to which the infeed may be exceeded.

Cycle parameters

Help graphic	Parameter
	Q460 Set-up clearance? Retraction before and after a cut. And distance for the pre-positioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	**Q558 Extensn. angle at contour start?** Angle in the WPL-CS, by which the cycle extends the contour up to the workpiece blank at the programmed starting point. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	**Q559 Extension angle at contour end?** Angle in WPL CS by which the cycle extends the contour at the programmed end point up to the workpiece blank. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	**Q478 Roughing feed rate?** Feed rate during roughing in millimeters per minute Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging Feed rate in millimeters per minute for plunging. This input value is optional. If you do not program the feed rate for plunging, the roughing feed rate **Q478** will apply. Input: **0...99999.999** or **FAUTO**
	Q556 Minimum angle of inclination? Smallest possible permitted angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q557 Maximum angle of inclination? Largest possible angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q567 Finishing allowance of contour? Contour-parallel oversize that will remain after roughing. This value has an incremental effect. Input: **-9...99.999**

Help graphic	Parameter
	**Q519 Infeed on contour?** Axial, radial and contour-parallel infeed (per cut). Enter a value greater than 0. This value has an incremental effect. Input: **0.001...99.999**
	Q463 Maximum cutting depth? Limit of the maximum infeed relative to the cutting edge. Depending on the tool angle of inclination, the control may temporarily exceed the **Q519 INFEED** (for example, when machining a corner). Use this optional parameter to limit the extent by which the infeed may be exceeded. If you define the value 0, the maximum infeed is two thirds of the length of the cutting edge. Input: **0...99.999**
	**Q590 Machining mode (0/1/2/3/4/5)?** Defining the direction of machining: **0**: Automatic; the control automatically combines transverse and longitudinal machining. **1**: Longitudinal turning (outside) **2**: Face turning (front face) **3**: Longitudinal turning (inside) **4**: Face turning (chuck) **5**: Contour-parallel Input: **0**, **1**, **2**, **3**, **4**, **5**
	Q591 Machining sequence (0/1)? Define the machining sequence after which the control executes the contour: **0**: Machining occurs in segments. The sequence is selected in such a way that the center of gravity of the workpiece is shifted towards the chuck as soon as possible. **1**: The workpiece is machined paraxially. The sequence is selected in such a way that the moment of inertia of the workpiece decreases as soon as possible. Input: **0**, **1**
	Q389 Machining strategy (0/1)? Definite the cutting direction: **0**: Unidirectional; every cut is made in the direction of the contour. The direction of the contour depends on **Q499** **1**: Bidirectional; cuts are made against the direction of the contour. The cycle determines the best direction for each following step. Input: **0**, **1**

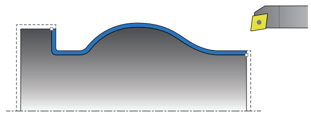
Example

```
11 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q499=+0                ;REVERSE CONTOUR ~
    Q558=+0                ;EXT:ANGLE CONT.START ~
    Q559=+90               ;CONTOUR END EXT ANGL ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q488=+0.3              ;PLUNGING FEED RATE ~
    Q556=+0                ;MIN. INCLINAT. ANGLE ~
    Q557=+90               ;MAX. INCLINAT. ANGLE ~
    Q567=+0.4              ;FINISH. ALLOW. CONT. ~
    Q519=+2                ;INFEED ~
    Q463=+3                ;MAX. CUTTING DEPTH ~
    Q590=+0                ;MACHINING MODE ~
    Q591=+0                ;MACHINING SEQUENCE ~
    Q389=+1                ;UNI.- BIDIRECTIONAL
12 L  X+58  Y+0 FMAX M303
13 L  Z+50 FMAX
14 CYCL CALL
```

14.34 Cycle 883 TURNING SIMULTANEOUS FINISHING (option 158)

ISO programming
G883

Application



Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.
The cycle is machine-dependent.

You can use this cycle to machine complex contours that are only accessible with different inclinations. When machining with this cycle, the inclination between tool and workpiece changes. This results in machining operations with at least three axes (two linear axes and one rotary axis).

The cycle monitors the workpiece contour with respect to the tool and the tool carrier. The cycle avoids unnecessary tilting movements in order to machine optimum surfaces.

If you want to force tilting movements, you can define inclination angles at the beginning and at the end of the contour. Even if simple contours have to be machined, you can use a large area of the indexable insert to achieve longer tool life.

Execution with a FreeTurn tool

You can execute this cycle with FreeTurn tools. This method allows you to perform the most common turning operations with just one tool. Machining times can be reduced through the flexible tool because fewer tool changes occur.

Requirements:

- This function must be adapted by your machine manufacturer.
- You must properly define the tool.

The NC program remains unchanged except for the calling of the FreeTurn cutting edges, see "Example: Turning with a FreeTurn tool", Page 715

Finishing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control moves the tool to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 If programmed, the tool traverses to the inclination angle that was calculated by the control based on the minimum and maximum inclination angles you have defined.
3 The control finishes the contour of the finished part (contour starting point to contour end point) simultaneously at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The cycle uses the position of the tool at cycle call as the cycle starting position. Incorrect pre-positioning can cause contour damage. There is a danger of collision!

- Move the tool to a safe position in the X and Z axes.

NOTICE

Danger of collision!

If the contour ends too closely at the fixture, a collision between tool and fixture might occur during machining.

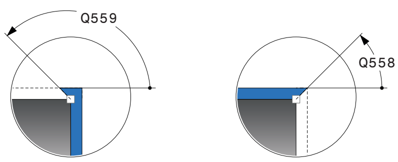
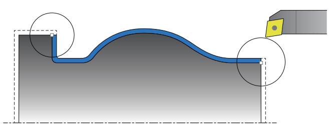
- When clamping, take both the tool angle of inclination and the departure movement into account

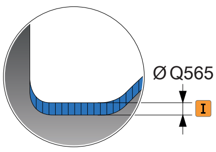
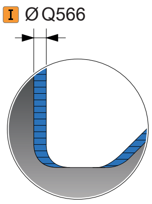
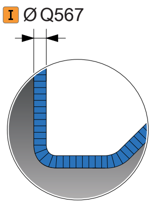
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- Based on the programmed parameters, the control calculates only **one** collision-free path.
- Software limit switches limit the possible inclination angles **Q556** and **Q557**. If, in **Test Run** the software limit switches are deactivated, then the simulation may deviate from the later machining operation.
- The cycle calculates a collision-free path. For this purpose, it only uses the 2D contour of the tool holder without considering the Y axis depth.

Notes on programming

- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- Move the tool to a safe position before the cycle call.
- The cycle requires a radius compensation (**RL**/**RR**) in its contour description.
- Prior to the cycle call, you must program **FUNCTION TCPM**. HEIDENHAIN recommends programming the tool reference point **REFPNT TIP-CENTER** in **FUNCTION TCPM**. Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip.
  Further information: User's Manual for **Klartext Programming**
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Please note: The smaller the resolution in cycle parameter **Q555** is, the easier will it be to find a solution even in complex situations. The drawback is that the calculation will take more time.
- For determining the inclination angle, the cycle requires the definition of a tool holder. For this purpose, assign a tool holder to the tool in the **KINEMATIC** column of the tool table.
- Please note that cycle parameters **Q565** (Finishing allowance in diameter) and **Q566** (Finishing allowance in Z) cannot be combined with **Q567** (Finishing allowance of contour)!

Cycle parameters

Help graphic	Parameter
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	**Q558 Extensn. angle at contour start?** Angle in the WPL-CS, by which the cycle extends the contour up to the workpiece blank at the programmed starting point. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	**Q559 Extension angle at contour end?** Angle in WPL CS by which the cycle extends the contour at the programmed end point up to the workpiece blank. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q556 Minimum angle of inclination? Smallest possible permitted angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q557 Maximum angle of inclination? Largest possible angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q555 Stepping angle for calculation? Cutting width for the calculation of possible solutions Input: **0.5...9.99**

Help graphic	Parameter
	Q537 Inclin. angle (0=N/1=J/2=S/3=E)? Define whether an inclination angle is active: **0**: No inclination angle active **1**: Inclination angle active **2**: Inclination angle at contour start active **3**: Inclination angle at contour end active Input: **0**, **1**, **2**, **3**
	Q538 Inclin. angle at contour start? Inclination angle at the beginning of the programmed contour (WPL-CS) Input: **-180...+180**
	Q539 Inclinatn. angle at contour end? Inclination angle at the end of the programmed contour (WPL-CS) Input: **-180...+180**
	**Q565 Finishing allowance in diameter?** Diameter oversize that remains on the contour after finishing. This value has an incremental effect. Input: **-9...99.999**
	**Q566 Finishing allowance in Z?** Oversize on the defined contour in the axial direction that remains on the contour after finishing. This value has an incremental effect. Input: **-9...99.999**
	**Q567 Finishing allowance of contour?** Contour-parallel oversize on the defined contour that remains after finishing. This value has an incremental effect. Input: **-9...99.999**

Example

```
11 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q499=+0              ;REVERSE CONTOUR ~
    Q558=+0              ;EXT:ANGLE CONT.START ~
    Q559=+90             ;CONTOUR END EXT ANGL ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q556=-30             ;MIN. INCLINAT. ANGLE ~
    Q557=+30             ;MAX. INCLINAT. ANGLE ~
    Q555=+7              ;STEPPING ANGLE ~
    Q537=+0              ;INCID. ANGLE ACTIVE ~
    Q538=+0              ;INCLIN. ANGLE START ~
    Q539=+0              ;INCLINATN. ANGLE END ~
    Q565=+0              ;FINISHING ALLOW. D. ~
    Q566=+0              ;FINISHING ALLOW. Z ~
    Q567=+0              ;FINISH. ALLOW. CONT.
12 L  X+58  Y+0 FMAX M303
13 L  Z+50 FMAX
14 CYCL CALL
```

14.35 Programming example

Example: Gear hobbing

The following NC program uses Cycle **880 GEAR HOBBING** This programming example illustrates the machining of a helical gear, with Module=2.1.

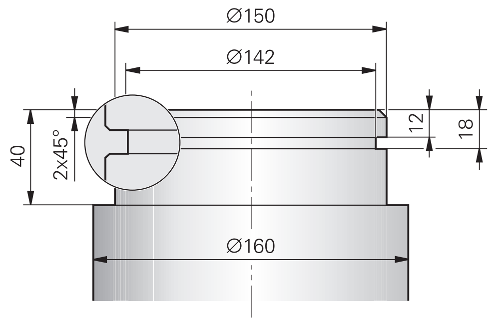
Program sequence

- Tool call: Gear hob
- Start turning mode
- Move to safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGIN PGM 8 MM	
1 BLK FORM CYLINDER Z R42 L150	
2 FUNCTION MODE MILL	; Activate milling mode
3 TOOL CALL "GEAD_HOB"	; Call tool
4 FUNCTION MODE TURN	; Activate turning mode
5 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
6 M145	; Cancel a potentially still active M144
7 FUNCTION TURNDATA SPIN VCONST:OFF S50	; Constant cutting speed OFF
8 M140 MB MAX	; Retract the tool
9 L A+0 R0 FMAX	; Set turning axis to 0
10 L X+250 Y-250 R0 FMAX M303	; Pre-position the tool in the working plane on the side on which machining will be performed, Spindle ON
11 L Z+20 R0 FMAX	; Pre-position the tool in the spindle axis
12 M136	; Feed rate in mm/rev.
13 CYCL DEF 880 GEAR HOBBING ~	
Q215=+0 ;MACHINING OPERATION ~	
Q540=+2.1 ;MODULE ~	
Q541=+0 ;NUMBER OF TEETH ~	
Q542=+69.3 ;OUTSIDE DIAMETER ~	
Q543=+0.1666 ;TROUGH-TIP CLEARANCE ~	
Q544=-5 ;ANGLE OF INCLINATION ~	
Q545=+1.6833 ;TOOL LEAD ANGLE ~	
Q546=+3 ;CHANGE TOOL DIRECTN. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+0 ;MACHINING SIDE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	
Q253=+800 ;F PRE-POSITIONING ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q553=+10 ;TOOL LENGTH OFFSET ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-10 ;END POINT IN Z ~	

Q463=+1 ;MAX. CUTTING DEPTH ~	
Q460=2 ;SAFETY CLEARANCE ~	
Q488=+1 ;PLUNGING FEED RATE ~	
Q478=+2 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q505=+1 ;FINISHING FEED RATE	
14 CYCL CALL	; Call cycle
15 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
16 M145	; Switch off active M144 in the cycle
17 FUNCTION MODE MILL	; Activate milling mode
18 M140 MB MAX	; Retract tool in the tool axis
19 L A+0 C+0 R0 FMAX	; Reset turning
20 M30	; End of program
21 END PGM 8 MM	

Example: Shoulder with recess



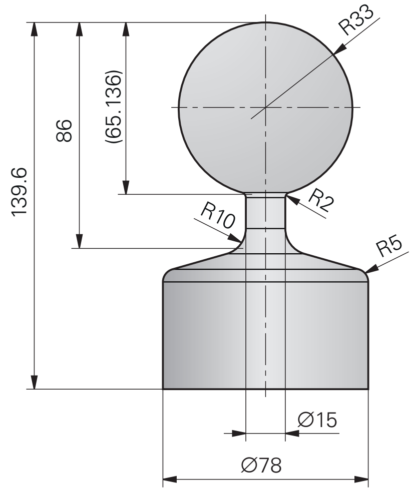
0 BEGIN PGM 9 MM	
1 BLK FORM CYLINDER Z R80 L60	
2 TOOL CALL 301	; Tool call
3 M140 MB MAX	; Retract the tool
4 FUNCTION MODE TURN	; Activate turning mode
5 FUNCTION TURNDATA SPIN VCONST:ON VC:150	; Constant cutting speed
6 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532=+750 ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
7 M136	; Feed rate in mm/rev.
8 L X+165 Y+0 R0 FMAX	; Approach starting point in the plane
9 L Z+2 R0 FMAX M304	; Safety clearance, turning spindle on
10 CYCL DEF 812 SHOULDER, LONG. EXT. ~	
Q215=+0 ;MACHINING OPERATION ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q491=+160 ;DIAMETER AT CONTOUR START ~	
Q492=+0 ;CONTOUR START IN Z ~	
Q493=+150 ;DIAMETER AT CONTOUR END ~	
Q494=-40 ;CONTOUR END IN Z ~	
Q495=+0 ;ANGLE OF CIRCUM. SURFACE ~	
Q501=+1 ;TYPE OF STARTING ELEMENT ~	
Q502=+2 ;SIZE OF STARTING ELEMENT ~	
Q500=+1 ;RADIUS OF CONTOUR EDGE ~	
Q496=+0 ;ANGLE OF FACE ~	
Q503=+1 ;TYPE OF END ELEMENT ~	
Q504=+2 ;SIZE OF END ELEMENT ~	
Q463=+2.5 ;MAX. CUTTING DEPTH ~	
Q478=+0.25 ;ROUGHING FEED RATE ~	

Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q484=+0.2 ;OVERSIZE IN Z ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q506=+0 ;CONTOUR SMOOTHING	
11 CYCL CALL	; Cycle call
12 M305	; Turning spindle off
13 TOOL CALL 307	; Tool call
14 M140 MB MAX	; Retract the tool
15 FUNCTION TURNDATA SPIN VCONST:ON VC:100	; Constant cutting speed
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532=+750 ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+0 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
17 L X+165 Y+0 R0 FMAX	; Approach starting point in the plane
18 L Z+2 R0 FMAX M304	; Safety clearance, turning spindle on
19 CYCL DEF 862 EXPND. RECESS, RADL. ~	
Q215=+0 ;MACHINING OPERATION ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q491=+150 ;DIAMETER AT CONTOUR START ~	
Q492=-12 ;CONTOUR START IN Z ~	
Q493=+142 ;DIAMETER AT CONTOUR END ~	
Q494=-18 ;CONTOUR END IN Z ~	
Q495=+0 ;ANGLE OF SIDE ~	
Q501=+1 ;TYPE OF STARTING ELEMENT ~	
Q502=+1 ;SIZE OF STARTING ELEMENT ~	
Q500=+0 ;RADIUS OF CONTOUR EDGE ~	
Q496=+0 ;ANGLE OF SECOND SIDE ~	
Q503=+1 ;TYPE OF END ELEMENT ~	
Q504=+1 ;SIZE OF END ELEMENT ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q484=+0.2 ;OVERSIZE IN Z ~	
Q505=+0.15 ;FINISHING FEED RATE ~	
Q463=+0 ;LIMIT TO DEPTH ~	
Q510=+0.8 ;RECESSING OVERLAP ~	
Q511=+80 ;FEED RATE FACTOR ~	
Q462=+0 ;RETRACTION MODE ~	
Q211=+3 ;DWELL TIME IN REVS ~	

Q562=+1 ;MULTIPLE PLUNGING	
20 CYCL CALL M8	; Cycle call
21 M305	; Turning spindle off
22 M137	; Feed rate in mm/minute
23 M140 MB MAX	; Retract the tool
24 FUNCTION MODE MILL	; Activate milling mode
25 M30	; End of program
26 END PGM 9 MM	

Example: Simultaneous turning

The following NC program uses Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** and Cycle **883 TURNING SIMULTANEOUS FINISHING**.



Program sequence

- Call the tool (e.g., TURN_ROUGH)
- Activate turning mode
- Pre-position
- Select the contours by using **SEL CONTOUR**
- Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Call the cycle
- Tool call (e.g., TURN_FINISH)
- Activate turning mode
- Cycle **883 TURNING SIMULTANEOUS FINISHING**
- Call the cycle
- End of program

0 BEGIN PGM 1341941_1 MM	
1 BLK FORM ROTATION Z DIM_D FILE "1341941_blank.H"	
2 FUNCTION MODE TURN	; Activate turning mode
3 TOOL CALL "TURN_ROUGH"	; Tool call
4 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+1 ;ANGLE OF INCIDENCE ~	
Q532=MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
5 FUNCTION TURNDATA SPIN VCONST: ON VC:400 SMAX800	; Constant surface speed
6 M145	; Reset the tool offset
7 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; Activate TCPM
8 L X+120 Y+0 R0 FMAX	; Pre-position
9 L Z+20 R0 FMAX M303	
10 FUNCTION TURNDATA BLANK "1341941_blank.H"	; Workpiece blank update
11 SEL CONTOUR "1341941_finish.h"	; Define the contour
12 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=-90 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	

Q478=+0.3 ;ROUGHING FEED RATE ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	
Q556=-80 ;MIN. INCLINAT. ANGLE ~	
Q557=+90 ;MAX. INCLINAT. ANGLE ~	
Q567=+0.4 ;FINISH. ALLOW. CONT. ~	
Q519=+2 ;INFEED ~	
Q463=+2.5 ;MAX. CUTTING DEPTH ~	
Q590=+1 ;MACHINING MODE ~	
Q591=+0 ;MACHINING SEQUENCE ~	
Q389=+0 ;UNI.- BIDIRECTIONAL	
13 CYCL CALL	; Cycle call
14 M305	
15 TOOL CALL "TURN_FINISH"	; Tool call
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+1 ;ANGLE OF INCIDENCE ~	
Q532=MAX ;FEED RATE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
17 FUNCTION TURNDATA SPIN VCONST: ON VC:400 SMAX800	; Constant surface speed
18 M145	; Reset the tool offset
19 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; Activate TCPM
20 L X+120 Y+0 R0 FMAX	
21 L Z+20 R0 FMAX M303	
22 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=-90 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q556=-80 ;MIN. INCLINAT. ANGLE ~	
Q557=+90 ;MAX. INCLINAT. ANGLE ~	
Q555=+1 ;STEPPING ANGLE ~	
Q537=+0 ;INCID. ANGLE ACTIVE ~	
Q538=+0 ;INCLIN. ANGLE START ~	
Q539=+0 ;INCLINATN. ANGLE END ~	
Q565=+0 ;FINISHING ALLOW. D. ~	

Q566=+0 ;FINISHING ALLOW. Z ~	
Q567=+0 ;FINISH. ALLOW. CONT.	
23 CYCL CALL	; Cycle call
24 M305	
25 FUNCTION TURNDATA BLANK OFF	; Deactivate workpiece blank update
26 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
27 FUNCTION MODE MILL	; Activate milling mode
28 TOOL CALL 0 Z	
29 PLANE RESET TURN FMAX	
30 M30	; End of program
31 END PGM 1341941_1 MM	

NC program 1341941_blank.h

```
0  BEGIN PGM 1341941_BLANK MM
1  L  X+0  Z+0.4
2  L  X+80
3  L  Z-139.6
4  L  X+0
5  L  Z+0.4
6  END PGM 1341941_BLANK MM
```

NC program 1341941_finish.h

```
0  BEGIN PGM 1341941_FINISH MM
1  L  X+0  Z+0 RR
2  CR  Z-65.136  X+15 R+33 DR+
3  RND R2
4  L  Z-86
5  RND R10
6  L  X+78 Z-95
7  RND R5
8  L  Z-100
9  END PGM 1341941_FINISH MM
```

Example: Turning with a FreeTurn tool

Cycles **882 SIMULTANEOUS ROUGHING FOR TURNING** and **883 TURNING SIMULTANEOUS FINISHING** are used in the following NC program.

Program sequence:

- Activate turning mode
- Call FreeTurn tool with second cutting edge
- Adjust the coordinate system with cycle **800 ADJUST XZ SYSTEM**
- Move to safe position
- Call cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Call FreeTurn tool with second cutting edge
- Move to safe position
- Call cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Move to safe position
- Call cycle **883 TURNING SIMULTANEOUS FINISHING**
- Reset active transformation with the PC program **RESET.h**

0 BEGIN PGM FREETURN MM	
1 FUNCTION MODE TURN "AC_TURN"	; Activate turning mode
2 PRESET SELECT #16	
3 BLK FORM CYLINDER Z D100 L101 DIST+1	
4 FUNCTION TURNDATA BLANK LBL 1	; Activate blank form update
5 TOOL CALL 145.0	; Call FreeTurn tool with first edge
6 M136	
7 FUNCTION TURNDATA SPIN VCONST:ON VC:250	; Constant cutting speed
8 L Z+50 R0 FMAX M303	
9 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+90 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
10 CYCL DEF 14.0 CONTOUR	
11 CYCL DEF 14.1 CONTOUR LABEL2	
12 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=+0 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	

Q488=+0.3 ;PLUNGING FEED RATE ~	
Q556=+30 ;MIN. INCLINAT. ANGLE ~	
Q557=+160 ;MAX. INCLINAT. ANGLE ~	
Q567=+0.3 ;FINISH. ALLOW. CONT. ~	
Q519=+2 ;INFEED ~	
Q463=+2 ;MAX. CUTTING DEPTH ~	
Q590=+5 ;MACHINING MODE ~	
Q591=+1 ;MACHINING SEQUENCE ~	
Q389=+0 ;UNI.- BIDIRECTIONAL	
13 L X+105 Y+0 R0 FMAX	
14 L Z+2 R0 FMAX M99	
15 TOOL CALL 145.1	; Call FreeTurn tool with second cutting edge
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+90 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
17 Q519 = 1	; Reduce infeed to 1
18 L X+105 Y+0 R0 FMAX	; Approach starting point
19 L Z+2 R0 FMAX M99	; Call cycle
20 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=+0 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q556=+30 ;MIN. INCLINAT. ANGLE ~	
Q557=+160 ;MAX. INCLINAT. ANGLE ~	
Q555=+5 ;STEPPING ANGLE ~	
Q537=+0 ;INCID. ANGLE ACTIVE ~	
Q538=+90 ;INCLIN. ANGLE START ~	
Q539=+0 ;INCLINATN. ANGLE END ~	
Q565=+0 ;FINISHING ALLOW. D. ~	
Q566=+0 ;FINISHING ALLOW. Z ~	
Q567=+0 ;FINISH. ALLOW. CONT.	
21 L X+105 Y+0 R0 FMAX	; Approach starting point
22 L Z+2 R0 FMAX M99	; Call cycle
23 CALL PGM RESET.H	; Call **RESET** program

24 M30	; End of program
25 LBL 1	; Define **LBL 1**
26 L X+100 Z+1	
27 L X+0	
28 L Z-60	
29 L X+100	
30 L Z+1	
31 LBL 0	
32 LBL 2	; Define **LBL 2**
33 L Z+1 X+60 RR	
34 L Z+0	
35 L Z-2 X+70	
36 RND R2	
37 L X+80	
38 RND R2	
39 L Z+0 X+98	
40 RND R2	
41 L Z-10	
42 RND R2	
43 L Z-8 X+89	
44 RND R2	
45 L Z-15 X+60	
46 RND R2	
47 L Z-55	
48 RND R2	
49 L Z-50 X+98	
50 RND R2	
51 L Z-60	
52 LBL 0	
53 END PGM FREETURN MM	

15

Cycles: Grinding

15.1 Grinding cycles: general information

Overview

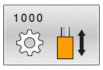
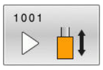
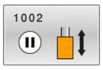
To define grinding cycles:

- Press the **CYCL DEF** key

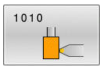
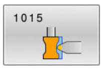
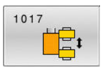
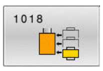
- Press the **GRINDING** soft key
- Select the cycle group (e.g., cycles for dressing)
- Select the desired cycle (e.g., **DRESSING DIAMETER**).

The control offers the following cycles for grinding operations:

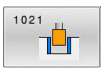
Reciprocating strokes

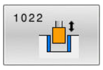
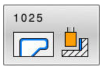
Soft key	Cycle	Page
	Cycle 1000 DEFINE RECIP. STROKE (option 156) ■ Define the reciprocating stroke and start it, if applicable	722
	Cycle 1001 START RECIP. STROKE (option 156) ■ Start reciprocating stroke	725
	Cycle 1002 STOP RECIP. STROKE (option 156) ■ Stop the reciprocating stroke and clear it, if applicable	726

Dressing

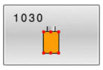
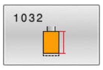
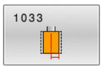
Soft key	Cycle	Page
	Cycle 1010 DRESSING DIAMETER (option 156) ■ Dressing a grinding wheel diameter	731
	Cycle 1015 PROFILE DRESSING (option 156) ■ Dressing a defined grinding wheel profile	736
	Cycle 1016 DRESSING OF CUP WHEEL (option 156) ■ Dressing a cup wheel	742
	Cycle 1017 DRESSING WITH DRESSING ROLL (option 156) ■ Dressing with a dressing roll ■ Reciprocating strokes ■ Oscillating ■ Fine oscillating	747
	Cycle 1018 RECESSING WITH DRESSING ROLL (option 156) ■ Dressing with a dressing roll ■ Recessing ■ Multiple recessing	754

Grinding

Soft key	Cycle	Page
	Cycle 1021 CYLINDER, SLOW-STROKE GRINDING (option 156) ■ Grinding inside or outside cylindrical contours ■ Multiple circular paths during a reciprocating stroke	760

Soft key	Cycle	Page
	Cycle 1022 CYLINDER, FAST-STROKE GRINDING (option 156) ■ Grinding inside or outside cylindrical contours ■ Grind with circular and helical paths, motion may have superimposed reciprocating stroke	768
	Cycle 1025 GRINDING CONTOUR (option 156) ■ Grinding open and closed contours	774

Special cycles

Soft key	Cycle	Page
	Cycle 1030 ACTIVATE WHEEL EDGE (option 156) ■ Activating the desired wheel edge	778
	Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156) ■ Compensation of the length in absolute or incremental values	780
	Cycle 1033 GRINDING WHL RADIUS COMPENSATION (option 156) ■ Compensation of the radius in absolute or incremental values	782

General information on jig grinding

Jig grinding means grinding of a 2D contour. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin. Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Outline: Grinding with a reciprocating stroke

```
0 BEGIN PGM GRIND MM
1 FUNCTION MODE MILL
2 TOOL CALL "GRIND_1" Z S20000
3 CYCL DEF 1000 DEFINE RECIP. STROKE
...
4 CYCL DEF 1001 START RECIP. STROKE
...
5 CYCL DEF 14 CONTOUR
...
6 CYCL DEF 1025 GRINDING CONTOUR
...
7 CYCL CALL
8 CYCL DEF 1002 STOP RECIP. STROKE
...
9 END PGM GRIND MM
```

15.2 Cycle 1000 DEFINE RECIP. STROKE (option 156)

ISO programming
G1000

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1000 DEFINE RECIP. STROKE** to define a reciprocating stroke in the tool axis and start reciprocating. This movement is executed as a superimposed movement. Thus, it is possible to execute any positioning block in parallel to the reciprocating stroke, even in the axis that is reciprocating. Once you started the reciprocating stroke, you can call a contour and start grinding.

- If you set **Q1004** to **0**, no reciprocating stroke will take place. In this case, you only define the cycle. If required, call Cycle **1001 START RECIP. STROKE** later to start the reciprocating stroke
- If you set **Q1004** to **1**, the reciprocating stroke starts at the current position. Depending on the setting in **Q1002**, the control will start reciprocating the tool in the positive or negative direction first. This reciprocation movement will be superimposed on the programmed movements (X, Y, Z)

The following cycles can be called in combination with the reciprocating stroke:

- Cycle **24 SIDE FINISHING**
- Cycle **25 CONTOUR TRAIN**
- Cycles **25x POCKETS/STUDS/SLOTS**
- Cycle **276 THREE-D CONT. TRAIN**
- Cycle **274 OCM FINISHING SIDE**
- Cycle **1025 GRINDING CONTOUR**

- The control does not support mid-program startup while the reciprocating stroke is active.
- As long as the reciprocating stroke is active in the started NC program, you cannot switch to the **Manual operation** or **Positioning w/ Manual Data Input** operating mode.

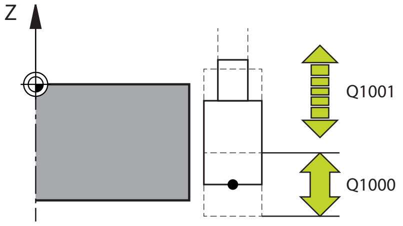
Notes

> Refer to your machine manual!
> The overrides for the reciprocation movements can be changed by the machine manufacturer.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1000** is DEF-active.
- The simulation of the superimposed movement can be seen in the **Program run, single block** and **Program run, full sequence** operating modes mode.
- Stop the reciprocating movement when you no longer need it. To do so, use **M30** or Cycle **1002 STOP RECIP. STROKE**. **STOP** or **M0** will not stop the reciprocating stroke.
- Reciprocating strokes can also be started in a tilted working plane. While the reciprocating stroke is active, however, you cannot change the orientation of the plane.
- You can also use a milling cutter with the superimposed reciprocating movement.

Cycle parameters

Help graphic | **Parameter**



Q1000 Length of reciprocating stroke?

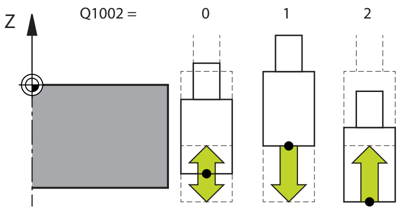
Length of the reciprocating movement, parallel to the active tool axis

Input: **0...9999.9999**

Q1001 Feed rate for reciprocation?

Speed of the reciprocating stroke in mm/min

Input: **0...999999**



Q1002 Type of reciprocation?

Definition of the start position. The direction of the first reciprocating stroke arises from this.

0: The current position is the middle of the stroke. The control first offsets the grinding tool by half the stroke in the negative direction and then continues the reciprocating movement in the positive direction

-1: The current position is the upper limit of the stroke. During the first stroke, the control offsets the grinding tool in the negative direction.

+1: The current position is the lower limit of the stroke. For the first stroke, the control offsets the grinding tool in the positive direction

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

Q1004 Start reciprocating stroke?

Definition of the effect of this cycle:

0: The reciprocating stroke is merely defined and may be started at a later time

+1: The reciprocating stroke is defined and started at the current position

Input: **0**, **1**

Example

```
11 CYCL DEF 1000 DEFINE RECIP. STROKE ~
    Q1000=+0              ;RECIPROCATING STROKE ~
    Q1001=+999            ;RECIP. FEED RATE ~
    Q1002=+1              ;RECIPROCATION TYPE ~
    Q1004=+0              ;START RECIP. STROKE
```

15.3 Cycle 1001 START RECIP. STROKE (option 156)

ISO programming

G1001

Application

> Refer to your machine manual.
>
> This function must be enabled and adapted by the machine manufacturer.

Cycle **1001 START RECIP. STROKE** starts a previously defined or stropped reciprocation movement. In an ongoing movement, this cycle has no effect.

Notes

> Refer to your machine manual!
>
> The overrides for the reciprocation movements can be changed by the machine manufacturer.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1001** is DEF-active.
- If you did not define a reciprocating stroke with Cycle **1000 DEFINE RECIP. STROKE**, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	Cycle **1001** does not have a cycle parameter. Conclude cycle input with the **END** key.

Example

```
11 CYCL DEF 1001 START RECIP. STROKE
```

15.4 Cycle 1002 STOP RECIP. STROKE (option 156)

ISO programming
G1002

Application

Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.

Cycle **1002 STOP RECIP. STROKE** stops the reciprocation movement. Depending on the setting in **Q1010**, the tool will stop immediately or traverse to its starting position.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1002** is DEF-active.

Note on programming

- Stopping the movement at the current position (**Q1010**=1) is allowed only if you simultaneously clear the definition of the reciprocating stroke (**Q1005**=1).

Cycle parameters

Help graphic	Parameter
	Q1005 Clear reciprocating stroke? Definition of the effect of this cycle: **0**: The reciprocating stroke is merely stopped and may be started again at a later time **+1**: The reciprocating stroke is stopped, and the definition of the reciprocating stroke from cycle **1000** is cleared Input: **0**, **1**
	Q1010 Stop reciproc. immediately (1)? Definition of the stopping position of the grinding tool: **0**: The stopping position is the same as the starting position **+1**: The stopping position is the same as the current position Input: **0**, **1**

Example

```
11 CYCL DEF 1002 STOP RECIP. STROKE ~
    Q1005=+0              ;CLEAR RECIP. STROKE ~
    Q1010=+0              ;RECIP.STROKE STOPPOS
```

15.5 General information on the dressing cycles

Fundamentals

> Refer to your machine manual.
> For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.

The term "dressing" refers to the sharpening or truing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, in dressing, the grinding tool is the workpiece.

The dressing operation removes material from the grinding wheel and may cause wear of the dressing tool. The material removal and wear lead to changed tool data that need to be compensated for after dressing.

The following dressing cycles are available:

- **1010 DRESSING DIAMETER**, see Page 731
- **1015 PROFILE DRESSING**, see Page 736
- **1016 DRESSING OF CUP WHEEL**, see Page 742
- **1017 DRESSING WITH DRESSING ROLL**, see Page 747
- **1018 RECESSING WITH DRESSING ROLL**, see Page 754

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge using Cycle **1030ACTIVATE WHEEL EDGE**.

Identify dressing operations in your NC program with **FUNCTION DRESS BEGIN / END**. When you activate **FUNCTION DRESS BEGIN**, the grinding wheel is redefined as the workpiece and the dressing tool as the tool. This might result in the axes moving in the opposite direction. When you terminate the dressing mode with **FUNCTION DRESS END**, the grinding wheel is redefined as the tool.

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

Structure of an NC program for dressing:

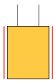
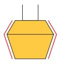
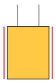
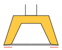
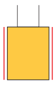
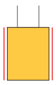
- Activate milling mode
- Call grinding wheel
- Move the tool to be dressed to a position near the dressing tool
- Activate dressing mode; select the kinematic model if necessary
- Activate wheel edge
- Call dressing tool; no mechanical tool change
- Call the cycle for dressing the diameter
- Deactivate dressing mode

```
0 BEGIN PGM GRIND MM
1 FUNCTION MODE MILL
2 TOOL CALL "GRIND_1" Z S20000
3 L X... Y... Z...
4 FUNCTION DRESS BEGIN
5 CYCL DEF 1030 ACTIVATE WHEEL EDGE
...
6 TOOL CALL "DRESS_1"
7 CYCL DEF 1010 DRESSING DIAMETER
...
8 FUNCTION DRESS END
9 END PGM GRIND MM
```

- The control does not support mid-program startup while dressing is active. If you jump to the first NC block after dressing using mid-program startup, the control will move the tool to the last position approached during dressing.

Dressing of grinding tools

The table below shows for each dressing cycle which grinding tools can be used with which dressing tools.

Cycle	Grinding tool	Dressing tool	Further Information
1010 **DRESSING DIAMETER**	Cylindrical grinding pin 	■ Stationary dresser with radius ■ Stationary dresser (flat) ■ Rotating dresser with radius ■ Rotating dresser (flat)	731
	Conical grinding pin 	■ Stationary dresser with radius ■ Stationary dresser (flat) ■ Rotating dresser with radius	
1015 **PROFILE DRESSING**	Cylindrical grinding pin 	■ Stationary dresser with radius ■ Stationary dresser (flat) ■ Rotating dresser with radius ■ Rotating dresser (flat)	736
1016 **DRESSING OF CUP WHEEL**	Cup wheel 	■ Stationary dresser with radius ■ Stationary dresser (flat) ■ Rotating dresser with radius	742
1017 **DRESSING WITH DRESSING ROLL**	Cylindrical grinding pin 	■ Rotating dresser (flat)	747
1018 **RECESSING WITH DRESSING ROLL**	Cylindrical grinding pin 	■ Rotating dresser (flat)	754

Notes

- Cycle **1010 DRESSING DIAMETER** can be used for dressing a diameter. If the grinding tool has corner radii, you cannot use dressing cycle **1010**. In this case, dressing would violate the radius shape. To enable dressing a diameter and a corner radius, dressing cycle **1015 PROFILE DRESSING** must be used.
- The control does not support mid-program startup while dressing is active. If you jump to the first NC block after dressing using mid-program startup, the control will move the tool to the last position approached during dressing.
- If you interrupt a dressing infeed movement, the last infeed will not be considered. If applicable, the dressing tool executes the first infeed or part of it without removing material if the dressing cycle is called again.
- Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.
- Please note that the switchover to dressing mode might have been programmed into the cycle sequence already by the machine manufacturer.

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

15.6 Cycle 1010 DRESSING DIAMETER (option 156)

ISO programming
G1010

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

Cycle **1010 DRESSING DIAMETER** allows you to dress the outside diameter of your grinding wheel. Depending on the strategy, the control causes movements based on the wheel geometry. If the dressing strategy in **Q1016** was set to 1 or 2, the path of the tool to the starting point is not along the grinding wheel, but via a retract path. The control does not apply tool radius compensation in the dressing cycle.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	1, 3, 5, 7	not supported

> If you work with the dressing roll tool type, then only the grinding pin is permitted.

Further information: "Dressing of grinding tools", Page 729

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 778

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in the **Program run, single block** or **Program run, full sequence** operating mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

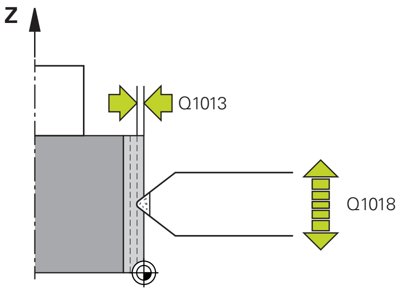
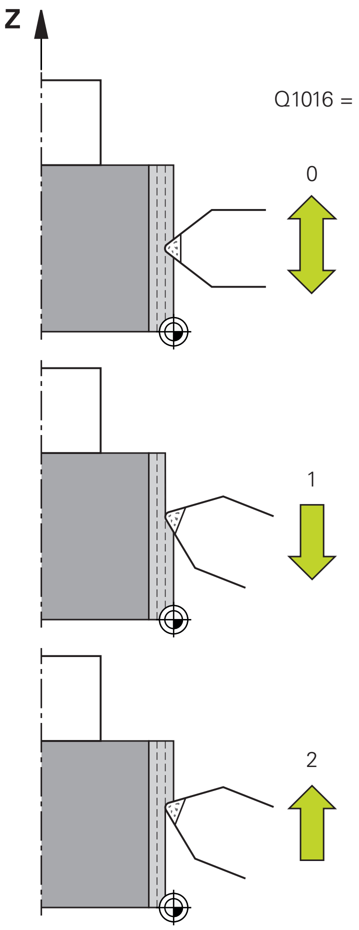
- Cycle **1010** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- The cycle supports dressing with a dressing role.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.
- Cycle **1010 DRESSING DIAMETER** can be used for dressing a diameter. If the grinding pin has corner radii, dressing would violate the radius shape. To enable dressing a diameter and corner radii, dressing cycle **1015 PROFILE DRESSING** must be used.

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

Information about dressing with a dressing role

- For the dressing tool, you must define the dressing role **TYPE**.
- For the dressing role, you must define a width: **CUTWIDTH**. The control takes the width into account during the dressing process.
- For dressing with a dressing role, only the dressing strategy **Q1016=0** is allowed.

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1016 Dressing strategy (0-2)?** Definition of the traversing movement during dressing: **0**: Reciprocating; dressing occurs in both directions **1**: Pulling; dressing occurs along the grinding wheel solely towards the active wheel edge **2**: Pushing; dressing occurs along the grinding wheel solely away from the active wheel edge Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via soft key. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1010 DRESSING DIAMETER ~
    Q1013=+0            ;DRESSING AMOUNT ~
    Q1018=+100          ;DRESSING FEED RATE ~
    Q1016=+1            ;DRESSING STRATEGY ~
    Q1019=+1            ;NUMBER INFEEDS ~
    Q1020=+0            ;IDLE STROKES ~
    Q1022=+0            ;COUNTER FOR DRESSING ~
    Q330=-1             ;TOOL ~
    Q1011=+0            ;FACTOR VC
```

15.7 Cycle 1015 PROFILE DRESSING (option 156)

ISO programming
G1015

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1015 PROFILE DRESSING** to dress a defined profile of your grinding wheel. The profile is defined in a profile program created as a separate NC program. This cycle is based on the grinding pin tool type. The starting point and end point of the profile must be identical (closed path) and are located at a corresponding position on the selected wheel edge. Define the return path to the starting point in your profile program. You must program the NC program in the ZX plane. Depending on the profile program, the control either does or does not use tool radius compensation. The activated wheel edge is used as the preset.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Dressing of grinding tools", Page 729

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 778

Cycle run

1 The control positions the dressing tool at the starting position with **FMAX**. The distance of the starting position from the datum is equal to the retraction values of the grinding wheel. The retraction values are relative to the active grinding edge.
2 The control offsets the datum to the extent of the dressing value and executes the profile program. This process repeats itself depending on the definition of **NUMBER INFEEDS Q1019**.
3 The control executes the profile program to the extent of the dressing value. If have programmed **NUMBER INFEEDS Q1019**, the infeeds repeat themselves. For every infeed, the dressing tool moves to the extent of the dressing value **Q1013**.
4 The profile program is repeated without infeed in accordance with **IDLE STROKES Q1020**.
5 The motion ends in the starting position.

- The datum of the workpiece system lies on the active wheel edge.

Description of function

Procedure for profile dressing

1. Defining the tool
   - Define the grinding tool in the tool table
   - Define the grinding tool type as grinding pin
2. Defining the NC program
   - Program the milling mode **FUNCTION MODE MILL**
   - Program the grinding tool call
   - Define Cycle **1030 ACTIVATE WHEEL EDGE**
   - Activate the dressing process with **FUNCTION DRESS BEGIN**
   - Program the dressing tool call

     The control does not exchange the active tool, but switches over by calculation.
   - Define cycle **1015 PROFILE DRESSING** and call up the profile program
   - Deactivate the dressing process with **FUNCTION DRESS END**
   - Program additional function **M30**
3. Creating the profile program
   - Program the desired profile as a contour

     The contour must be closed. The active edge is the profile datum. You program the traverse path.

     Further information: "Example of a profile program", Page 787

Applications for profile dressing

There are two applications for profile dressing:

- Shaping a grinding tool

  Further information: "Shaping a grinding tool", Page 738
- Resharpening a grinding tool

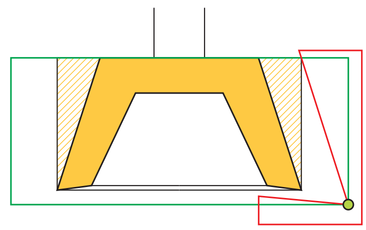
  Further information: "Resharpening a grinding tool", Page 738

In the examples below, a grinding pin is dressed to suit the profile of a cup wheel.

Shaping a grinding tool

If the grinding tool does not yet have the desired shape, it must be shaped.

The figure displays the following information:



Depiction	Definition
Yellow	Desired profile
Hatched	Finishing allowance from the grinding pin to the profile
Red line	Profile program
Green line	Diameter and length for the tool table
Green dot	Current grinding wheel edge

In order not to remove too much material in the first dressing process, the profile program must be relocated by at least the finishing allowance. The profile program datum can be relocated by enlarging the grinding tool radius and length in the tool table.

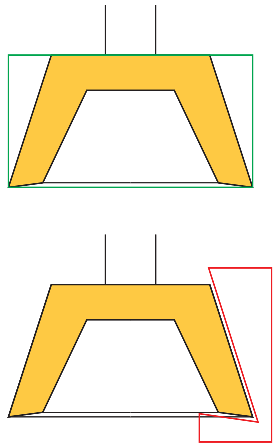
Define the grinding tool in the tool table to be so large that no part of the contour program will intersect the physical grinding tool.

> HEIDENHAIN recommends defining the grinding tool diameter and length large enough in the tool table!

The profile datum is the active edge that you define with Cycle **1030 ACTIVATE WHEEL EDGE**.

Resharpening a grinding tool

If the grinding tool already has the desired shape, you may resharpen it.



Depiction	Definition
Yellow	Desired profile
Red line	Profile program
Green line	Diameter and length for the tool table

The profile datum is the active edge that you define with Cycle **1030 ACTIVATE WHEEL EDGE**.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in the **Program run, single block** or **Program run, full sequence** operating mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

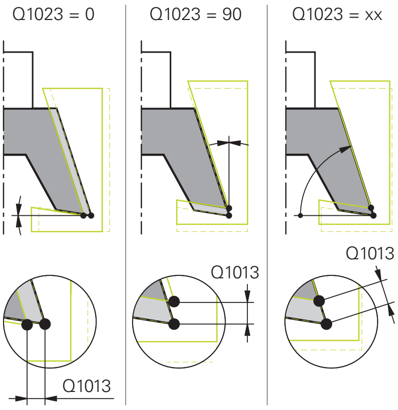
- Cycle **1015** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

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

Note on programming

- The angle of infeed must be selected in a way that the programmed profile always remains within the grinding wheel edge. If this condition is not met, then the dimensional accuracy of the grinding wheel is lost.

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1023 Infeed angle of profile program? Angle at which the profile of the program is moved into the grinding wheel. **0**: Infeed only at the diameter in the X axis of the dressing kinematic model **+90**: Infeed only in the Z axis of the dressing kinematic model Input: **0...90**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	Q1000 Name of the profile program? Enter the path and name of the NC program that will be used for the profile of the grinding wheel during the dressing process. Alternatively, select the profile program via the soft key **SELECT FILE**. Input: Max. **255** characters
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**

Help graphic	Parameter
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via soft key. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1015 PROFILE DRESSING ~
    Q1013=+0              ;DRESSING AMOUNT ~
    Q1023=+0              ;ANGLE OF INFEED ~
    Q1018=+100            ;DRESSING FEED RATE ~
    QS1000=""             ;PROFILE PROGRAM ~
    Q1019=+1              ;NUMBER INFEEDS ~
    Q1020=+0              ;IDLE STROKES ~
    Q1022=+0              ;COUNTER FOR DRESSING ~
    Q330=-1               ;TOOL ~
    Q1011=+0              ;FACTOR VC
```

15.8 Cycle 1016 DRESSING OF CUP WHEEL (option 156)

ISO programming
G1016

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1016 DRESSING OF CUP WHEEL** to dress the front face of a cup wheel. The activated wheel edge is used as the reference.

Depending on the strategy, the control causes movements based on the wheel geometry. If the dressing strategy in **Q1016** was set to **1** or **2**, the return of the tool to the starting point is not along the grinding wheel, but via a retract path.

If the Pull-and-Push strategy has been selected in dressing mode, the control will apply radius compensation. If the Reciprocating strategy has been selected in dressing mode, the control will not apply radius compensation.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
not supported	not supported	2, 6

Further information: "Dressing of grinding tools", Page 729

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 778

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in the **Program run, single block** or **Program run, full sequence** operating mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

The angle of inclination between the dressing tool and the cup wheel will not be monitored! There is a danger of collision!

- Make sure to program a dressing tool clearance angle greater than or equal to 0° relative to the front face of the cup wheel
- Verify the NC program by carefully executing it block by block

- Cycle **1016** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- The control saves the counter in the tool table. Its effect is global.

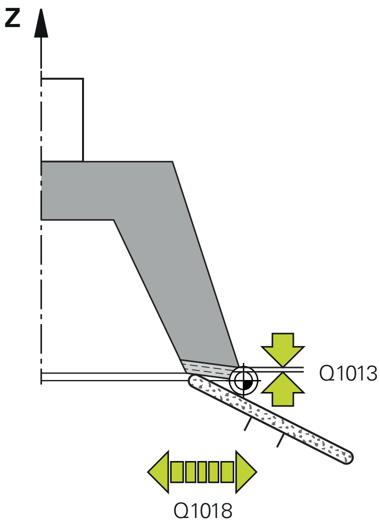
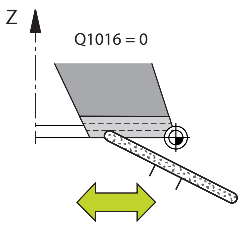
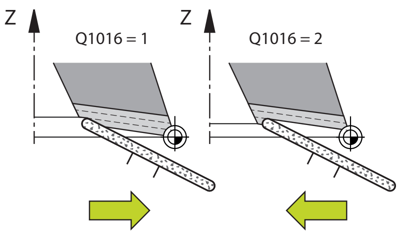
  Further information: User's Manual for Setup, Testing and Running NC Programs
- To enable dressing of the entire cutting edge, it is extended by twice the cutting-edge radius (2 x **RS**) of the dressing tool. Here, the minimum permissible radius (**R_MIN**) of the grinding wheel must not be undershot, otherwise the control interrupts the operation with an error message.
- In this cycle, the radius of the tool shank is not monitored.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

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

Notes on programming

- This cycle is permitted only for use with the cup wheel tool type. If you defined a different tool type, the control will display an error message.
- The strategy in **Q1016** = 0 (Reciprocating) is only possible for a straight front face angle (**HWA** = 0).

Cycle parameters

Help graphic	Parameter
  	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	Q1016 Dressing strategy (0-2)? Definition of the traversing movement during dressing: **0**: Reciprocating; dressing occurs in both directions **1**: Pulling; dressing occurs along the grinding wheel solely towards the active wheel edge **2**: Pushing; dressing occurs along the grinding wheel solely away from the active wheel edge Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via soft key. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1016 DRESSING OF CUP WHEEL ~
    Q1013=+0            ;DRESSING AMOUNT ~
    Q1018=+100          ;DRESSING FEED RATE ~
    Q1016=+1            ;DRESSING STRATEGY ~
    Q1019=+1            ;NUMBER INFEEDS ~
    Q1020=+0            ;IDLE STROKES ~
    Q1022=+0            ;COUNTER FOR DRESSING ~
    Q330=-1             ;TOOL ~
    Q1011=+0            ;FACTOR VC
```

15.9 Cycle 1017 DRESSING WITH DRESSING ROLL (option 156)

ISO programming
G1017

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

With cycle **1017 DRESSING WITH DRESSING ROLL**, you can dress the outside diameter of a grinding wheel with a dressing role. Depending on the dressing strategy, the control performs the appropriate movements in accordance with the wheel geometry.

The cycle offers the following dressing strategies:

- Reciprocating: lateral infeed at the reversal points of the reciprocating stroke
- Oscillating: interpolating infeed during a reciprocating stroke
- Fine Oscillating: interpolating infeed during a reciprocating stroke. After each interpolating infeed, a Z movement is performed without infeed in the dressing kinematic model.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Dressing of grinding tools", Page 729
Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 778

Cycle run

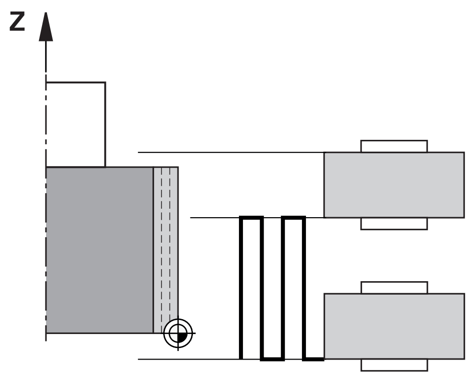
1 The control positions the dressing tool at the starting position with **FMAX**.
2 If you have defined a pre-position in **Q1025 PRE-POSITION**, the control approaches the position at **Q253 F PRE-POSITIONING**.
3 The control infeeds based on the dressing strategy.
Further information: "Dressing strategies", Page 748
4 After defining **IDLE STROKES** in **Q1020**, the control performs them after the last infeed.
5 The control moves to the starting position with **FMAX**.

Dressing strategies

> Depending on **Q1026 WEAR FACTOR**, the control divides the dressing value between the grinding wheel and the dressing roll.

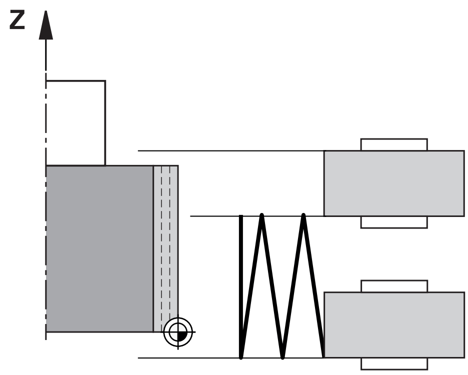
Reciprocating (**Q1024=0**)

1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The **DRESSING AMOUNT Q1013** is infed on the diameter at the **DRESSING FEED RATE Q1018**.
3 The control moves the dressing tool along the grinding wheel to the next reversal point of the reciprocating movement.
4 If other dressing infeeding is required, the control repeats processes 1 to 2 until the dressing process is complete.

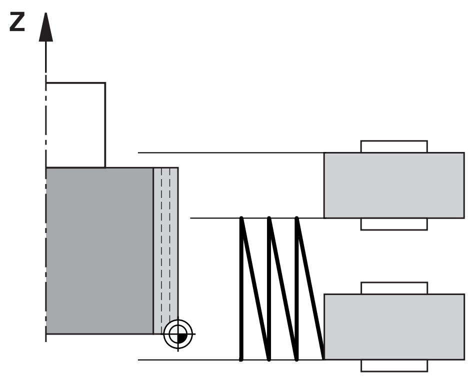


Oscillating (**Q1024=1**)

1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The control infeeds the **DRESSING AMOUNT Q1013** on the diameter. Infeeding is performed with interpolation at the dressing feed rate **Q1018** with the reciprocating stroke up to the next reversal point.
3 If there are more dressing infeed runs, then processes 1 to 2 are repeated until the dressing process is complete.
4 The control then retracts the tool without infeed in the Z axis of the dressing kinematic model to the other reversal point of the reciprocating movement.



Fine oscillating (**Q1024=2**)



1. The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2. The control infeeds the **DRESSING AMOUNT Q1013** on the diameter. Infeeding is performed with interpolation at the dressing feed rate **Q1018** with the reciprocating stroke up to the next reversal point.
3. The control then retracts the tool to the other reversal point of the reciprocating movement without an infeed cut.
4. If there is more infeeding, then processes 1 to 3 are repeated until the dressing procedure is complete.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in the **Program run, single block** or **Program run, full sequence** operating mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

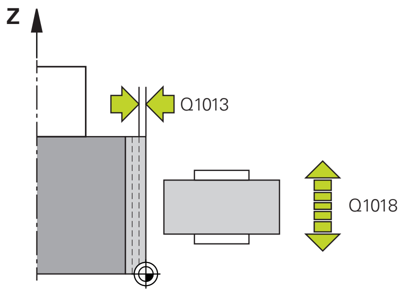
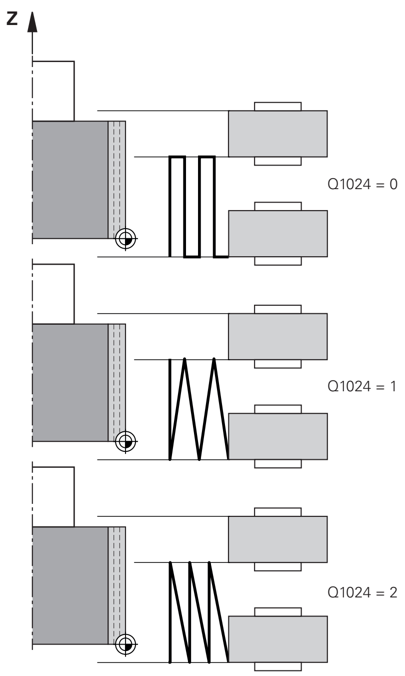
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

- Cycle **1017** is DEF-active.
- No coordinate conversion cycles are permitted in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, then the control performs the dressing process only after reaching the defined counter from the tool management function. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- At the end of each infeed run, the control updates the tool data for the grinding tool and dressing tool.
- For the reversal points of the reciprocating movement, the control takes into account the retraction values **AA** and **AI** from the tool management function. The width of the dressing roll must be less than the width of the dressing wheel, including the retraction values.
- The control does not apply tool radius compensation in the dressing cycle.

- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

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

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1024 Dressing strategy (0-2)?** Strategy during dressing with a dressing roll; **0**: Reciprocating; infeeding to the reversal points of the reciprocating motion. After the infeeding runs, the control executes a movement just in the Z axis within the dressing kinematic model. **1**: Oscillating; interpolated infeed during a reciprocating movement **2**: Fine oscillating; interpolated during a reciprocating movement. After every interpolated infeed run, the control executes a movement solely in the Z axis in the dressing kinematic model. Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1025 Distance for pre-positioning? Distance between the grinding wheel and the dressing role during pre-positioning Input: **0...9.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min. while approaching the pre-position Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q1026 Wear on dressing tool? Factor of the dressing value in order to define the wear on the dressing roll: **0**: The full dressing value is removed on the grinding wheel. **>0**: The factor is multiplied by the dressing value. The control takes the calculated value into account and assumes that this value will be lost during dressing due to wear on the dressing roll. The remaining dressing value is dressed on the grinding wheel. Input: **0...+0.99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via soft key. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1017 DRESSING WITH DRESSING ROLL ~
    Q1013=+0              ;DRESSING AMOUNT ~
    Q1018=+100            ;DRESSING FEED RATE ~
    Q1024=+0              ;DRESSING STRATEGY ~
    Q1019=+1              ;NUMBER INFEEDS ~
    Q1020=+0              ;IDLE STROKES ~
    Q1025=+5              ;PRE-POSITION DIST. ~
    Q253=+1000            ;F PRE-POSITIONING ~
    Q1026=+0              ;WEAR FACTOR ~
    Q1022=+2              ;COUNTER FOR DRESSING ~
    Q330=-1               ;TOOL ~
    Q1011=+0              ;FACTOR VC
```

15.10 Cycle 1018 RECESSING WITH DRESSING ROLL (option 156)

ISO programming
G1018

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

With Cycle **1018 RECESSING WITH DRESSING ROLL**, you can dress the outside diameter of a grinding wheel via recessing with dressing role. Depending on the dressing strategy, the control executes one or more recessing movements.

The cycle offers the following dressing strategies:

- **Recessing:** This strategy performs only linear recessing movements. The width of the dressing roll is larger than the dressing wheel width.
- **Multiple recessing:** This strategy executes linear recessing movements. At the end of the infeed run, the control moves the dressing tool in the Z axis of the dressing kinematic model and infeeds again.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Dressing of grinding tools", Page 729

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 778

Cycle run

Recessing

1. The control positions the dressing roll at the starting position at **FMAX**. At the starting position, the center of the dressing roll matches the middle of the grinding wheel edge. If **CENTER OFFSET Q1028** is programmed, then the control takes this into account when approaching the starting position.
2. The dressing roll approaches the **PRE-POSITION DIST. Q1025** at the feed rate **Q253 F PRE-POSITIONING**.
3. The dressing roll recesses into the grinding wheel with the **DRESSING FEED RATE Q1018** by the **DRESSING AMOUNT Q1013**.
4. If a **DWELL TIME IN REVS Q211** is defined, the control waits the defined amount of time.
5. The control retracts the dressing role with **F PRE-POSITIONING Q253** to the **PRE-POSITION DIST. Q1025**.
6. The control moves to the starting position with **FMAX**.

Multiple recessing

1. The control positions the dressing roll at the starting position with **FMAX**.
2. The dressing role approaches the **PRE-POSITION DIST. Q1025** at the feed rate **F PRE-POSITIONING Q253**.
3. The dressing roll recesses into the grinding wheel with the **DRESSING FEED RATE Q1018** by the **DRESSING AMOUNT Q1013**.
4. If a **DWELL TIME IN REVS Q211** is defined, then it is executed by the control.
5. At **F PRE-POSITIONING Q253**, the control retracts the dressing roll to the **PRE-POSITION DIST. Q1025**.
6. Based on the **RECESSING OVERLAP Q510**, the control moves the dressing roll to the next recessing position in the Z axis of the dressing kinematic model.
7. The control repeats processes 3 to 6 until the entire grinding wheel is dressed.
8. At **F PRE-POSITIONING Q253**, the control retracts the dressing role to the **PRE-POSITION DIST. Q1025**.
9. The control moves to the starting position at rapid traverse.

> The control calculates the number or required recesses based on the width of the grinding wheel, the width of the dressing roll and the value of the parameter **RECESSING OVERLAP Q510**.

Notes

NOTICE

Danger of collision!

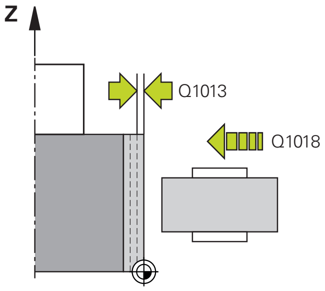
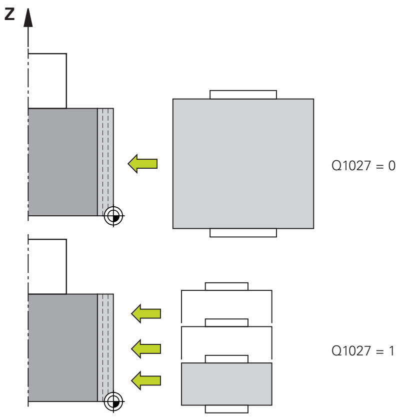
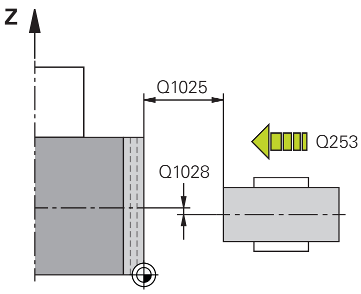
When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in the **Program run, single block** or **Program run, full sequence** operating mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

- Cycle **1018** is DEF-active.
- No coordinate transformations are allowed in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation.
- If the width of the dressing roll is less than the width of the grinding wheel, then use the dressing strategy multiple recessing **Q1027=1**.
- If you program a **COUNTER FOR DRESSING Q1022**, then the control performs the dressing process only after reaching the defined counter from the tool management function. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- At the end of every infeed run, the control corrects the tool data of the grinding tool and dressing tool.
- The control does not apply tool radius compensation in the dressing cycle.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

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

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1027 Dressing strategy (0-1)?** Strategy during recessing with a dressing roll: **0**: Recessing; the control executes a linear recessing movement. The grinding wheel width is less than the width of the dressing roll. **1**: Multiple recessing; the control executes linear recessing movements. After infeeding to the dressing value, the control moves the dressing tool in the Z axis in the dressing kinematic model and infeeds again. The width of the grinding wheel is greater than the width of the dressing roll. Input: **0**, **1**
	Q1025 Distance for pre-positioning? Distance between the grinding wheel and the dressing role during pre-positioning Input: **0...9.9999**
	**Q253 Feed rate for pre-positioning?** Traversing speed of the tool in mm/min. while approaching the pre-position Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q211 Dwell time / 1/min? Revolutions of the grinding wheel at the end of the recessing cut. Input: **0...999.99**
	Q1028 Offset of centers? Offset of the dressing roll center relative to the grinding wheel center. This offset takes effect in the Z axis of the dressing kinematic model. This value has an incremental effect. If **Q1027 = 1**, then the control does not use a center offset. Input: **-999.999...+999.999**

Help graphic	Parameter
	Q510 Overlap factor for recess width? With factor **Q510**, you influence the offset of the dressing roll in the Z axis of the dressing kinematic model. The control multiplies the factor with the value **CUTWIDTH** and offsets the dressing roll between the infeed runs by the calculated value. **1**: For every infeed run, the control recesses with the complete width of the dressing role. **Q510** takes effect only with **Q1027=1**. Input: **0.001...1**
	Q1026 Wear on dressing tool? Factor of the dressing value in order to define the wear on the dressing roll: **0**: The full dressing value is removed on the grinding wheel. **>0**: The factor is multiplied by the dressing value. The control takes the calculated value into account and assumes that this value will be lost during dressing due to wear on the dressing roll. The remaining dressing value is dressed on the grinding wheel. Input: **0...+0.99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via soft key. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1018 RECESSING WITH DRESSING ROLL ~
    Q1013=+1              ;DRESSING AMOUNT ~
    Q1018=+100            ;DRESSING FEED RATE ~
    Q1027=+0              ;DRESSING STRATEGY ~
    Q1025=+5              ;PRE-POSITION DIST. ~
    Q253=+1000            ;F PRE-POSITIONING ~
    Q211=+3               ;DWELL TIME IN REVS ~
    Q1028=+1              ;CENTER OFFSET ~
    Q510=+0.8             ;RECESSING OVERLAP~
    Q1026=+0              ;WEAR FACTOR ~
    Q1022=+2              ;COUNTER FOR DRESSING ~
    Q330=-1               ;TOOL ~
    Q1011=+0              ;FACTOR VC
```

15.11 Cycle 1021 CYLINDER, SLOW-STROKE GRINDING (option 156)

ISO programming
G1021

Application

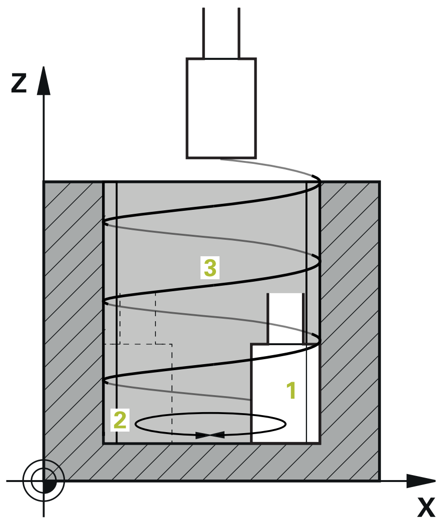
Refer to your machine manual!
This function must be enabled and adapted by the machine manufacturer.

Cycle **1021 CYLINDER, SLOW-STROKE GRINDING** allows you to grind circular pockets or circular studs. The height of the cylinder can be considerably greater than the width of the grinding wheel. Through a reciprocating stroke, the control can machine the complete height of the cylinder. The control executes multiple circular paths during the reciprocating stroke. In this process, the reciprocating stroke and the circular paths overlap to form a helix. This process is equivalent to grinding with a slow stroke.

The lateral infeed cuts occur at the reversal points of the reciprocating stroke along the semi-circle. You can program the feed rate of the reciprocating stroke as the pitch of the helical path relative to the width of the grinding wheel.

You can also completely machine cylinders without overshoot, such as blind holes. This is done by programming idle runs at the reversal points of the reciprocating stroke.

Cycle run



1 The control positions the grinding tool above the cylinder based on **POCKET POSITION Q367**. The control then moves the tool to the **CLEARANCE HEIGHT Q260** at rapid traverse.
2 The grinding tool uses **F PRE-POSITIONING Q253** for moving to the **SET-UP CLEARANCE Q200**
3 The grinding tool traverses to the starting point in the tool axis. The starting point depends on the **MACHINING DIRECTION Q1031**, upper or lower reversal point of the reciprocating stroke.
4 The cycle starts the reciprocating stroke. At the **GRINDING FEED RATE Q207**, the control moves the grinding tool to the contour.

   Further information: "Feed rate for the reciprocating stroke", Page 762
5 The control delays the reciprocating stroke in the starting position.
6 Depending on **Q1021 ONE-SIDED INFEED**, the control infeeds the grinding tool in a semi-circle around the lateral infeed **Q534** **1**.
7 As needed, the control executes the defined idle runs **2** **Q211** or **Q210**.

   Further information: "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 762
8 The cycle continues the reciprocating movement. The grinding tool follows multiple circular paths. The reciprocating stroke overlays the circular paths in the direction of the tool axis to form a helix. You can influence the pitch of the helical path by the factor **Q1032**.
9 The circular paths **3** repeat themselves until the second reversal point of the reciprocating stroke is reached.
10 The control repeats steps 4 to 7 until the diameter of the finished part **Q223** or the oversize **Q14** is reached.
11 After the last lateral infeed run, the grinding wheel moves the number of programmed idle strokes **Q1020** if applicable.
12 The control stops the reciprocating stroke. The grinding tool leaves the cylinder on a semi-circular path to the safety clearance **Q200**.
13 At **F PRE-POSITIONING Q253**, the grinding tool moves to the **SET-UP CLEARANCE Q200** and then at rapid traverse to the **CLEARANCE HEIGHT Q260**.

- In order for the grinding tool to completely machine the cylinder at the reversal points of the reciprocating stroke, you must define sufficient overshoot or idle runs.
- The length of the reciprocating stroke arises from the **DEPTH Q201**, the **SURFACE OFFSET Q1030** and the wheel width **B**.
- The starting point in the working plane is distant from the **FINISHED PART DIA. Q223** including **OVERSIZE AT START Q368** by the amount of the tool radius and the **SET-UP CLEARANCE Q200**.

Overshoot and idle runs to the reversal points of the reciprocating stroke

Path of the overshoot

Top	Bottom
This distance is defined in the parameter **Q1030 SURFACE OFFSET**.	You must add this distance to the machining depth and then define it in **Q201 DEPTH**.

If no overshoot is possible, such as with a pocket, program multiple idle runs at the reversal points of the reciprocating stroke (**Q210**, **Q211**). Select this number such that, after infeeding (half of a circular path), at least one circular path is traveled on the infed diameter. The number of idle runs is always based on a set feed-rate override of 100%.

- HEIDENHAIN recommends moving with a feed-rate override of 100% or more. A feed-rate override of less than 100% no longer ensures that the cylinder will be completely machined at the reversal points.
- For the definition of idle runs, HEIDENHAIN recommends defining at least a value of 1.5.

Feed rate for the reciprocating stroke

You can define the pitch per helical path (=360°) with the factor **Q1032**. Through this definition, the feed rate in mm or in inches/helical path (= 360°) can be derived for the reciprocating stroke.

The proportion of the **GRINDING FEED RATE Q207** to the feed rate of the reciprocating stroke plays a major role. If you deviate from a feed rate override of 100%, then ensure that the length of the reciprocating stroke during a circular path is less than the width of the grinding wheel.

HEIDENHAIN recommends selecting a factor of at most 0.5.

Notes

> The overrides for the reciprocation movements can be changed by the machine manufacturer.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The last lateral infeed may be smaller depending on the input.
- The control does not depict the reciprocating movement in the simulation. The reciprocating movement is depicted in the simulation graphics in the **Program run, single block** and **Program run, full sequence** operating modes.
- You can also execute this cycle with a milling cutter. In the case of a milling cutter, the tooth length **LCUTS** equals the width of the grinding wheel.
- Please note that the cycle takes **M109** into account. The **GRINDING FEED RATE Q207** in the status display during program run in the case of a pocket is therefore smaller than in the case of a stud. The control shows the feed rate of the center point path of the grinding tool, including the reciprocating stroke.

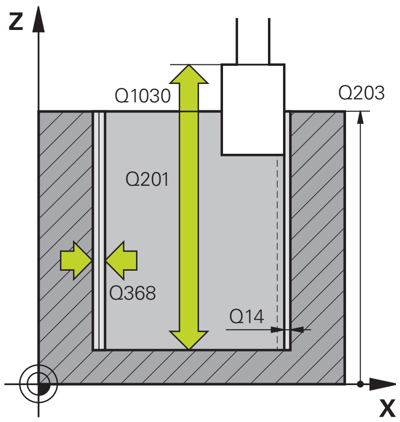
  Further information: User's Manual for Conversational Programming

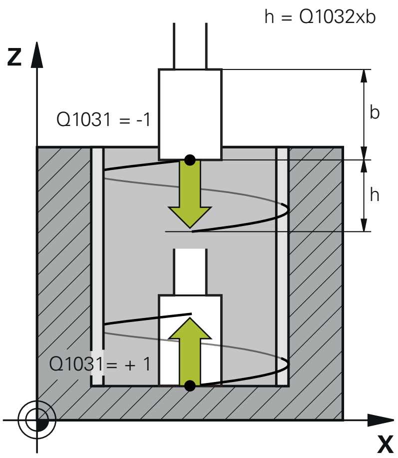
Notes on programming

- The control assumes that the bottom of the cylinder has a floor. For this reason, you can define an overshoot in **Q1030** only at the surface. If you machine a through hole, for example, then you must take into account the lower overshoot in **DEPTH Q201**.

  Further information: "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 762
- If the grinding wheel is wider than **DEPTH Q201** and the **SURFACE OFFSET Q1030**, then the control issues a **No swing stroke** error message. In this case, the resulting reciprocating stroke would be equal to 0.

Cycle parameters

Help graphic	Parameter
	Q650 Type of figure? Geometry of the figure: **0**: Pocket **1**: Island Input: **0**, **1**
	Q223 Finished part diameter? Diameter of the fully machined cylinder Input: **0...99999.9999**
	**Q368 Side oversize before machining?** Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+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**
	Q1030 Offset to surface? Position of the upper edge of the tool on the surface. The offset serves as the overshoot path on the surface for the reciprocating stroke. This value has an absolute effect. Input: **0...999.999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**

Help graphic	Parameter
	**Q1031 Machining direction?** Definition of the start position. The direction of the first reciprocating stroke arises from this. **−1** or **0**: The starting position is on the surface. The reciprocating stroke begins in the negative direction. **+1**: The starting position is at the cylinder floor. The reciprocating stroke begins in the positive direction. Input: **-1**, **0**, **+1**
	Q1021 One-sided infeed (0/1)? Position at which the lateral infeed occurs: **0**: Lower and upper lateral infeed **1**: One-sided infeed depending on **Q1031** ■ If **Q1031 = -1**, then the lateral infeed is performed above. ■ If **Q1031 = +1**, then the lateral infeed is performed below. Input: **0**, **1**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q1020 Number of idle strokes? Number of idle strokes after the last lateral infeed without material removal. Input: **0...99**
	Q1032 Factor for pitch of helix? The pitch per helical path (= 360°) arises from the factor **Q1032**. **Q1032** is multiplied by the width **B** of the grinding tool. The feed rate for the reciprocating stroke is influenced by the pitch of the helical path. **Further information:** "Feed rate for the reciprocating stroke", Page 762 Input: **0.000...1000**
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q15 Up-cut / climb grinding (-1/+1)? Define the type of contour grinding: **+1**: Climb grinding **-1** or **0**: Up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** 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**
	Q211 Idle runs at depth? Number of idle runs at the lower reversal point of the reciprocating stroke. **Further information:** "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 762. Input: **0...99.99**
	Q210 Idle runs at top? Number of idle runs at the upper reversal point of the reciprocating stroke. **Further information:** "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 762. Input: **0...99.99**

Example

```
11 CYCL DEF 1021 CYLINDER, SLOW-STROKE GRINDING ~
    Q650=+0              ;FIGURE TYPE ~
    Q223=+50             ;FINISHED PART DIA. ~
    Q368=+0.1            ;OVERSIZE AT START ~
    Q14=+0               ;ALLOWANCE FOR SIDE ~
    Q367=+0              ;POCKET POSITION ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q1030=+2             ;VERSATZ OBERFLAECHE ~
    Q201=-20             ;DEPTH ~
    Q1031=+1             ;MACHINING DIRECTION ~
    Q1021=+0             ;ONE-SIDED INFEED ~
    Q534=+0.01           ;LATERAL INFEED ~
    Q1020=+0             ;IDLE STROKES ~
    Q1032=+0.5           ;FAKTOR ZUSTELLUNG ~
    Q207=+2000           ;GRINDING FEED RATE ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q15=-1               ;TYPE OF GRINDING ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q211=+0              ;IDLE RUNS AT DEPTH ~
    Q210=+0              ;IDLE RUNS AT TOP
```

15.12 Cycle 1022 CYLINDER, FAST-STROKE GRINDING (option 156)

ISO programming

G1022

Application

> Refer to your machine manual!
> This function must be enabled and adapted by the machine manufacturer.

With the cycle **1022 CYLINDER, FAST STROKE GRINDING**, you can grind circular pockets and circular studs. In the process, the control executes circular and helical paths in order to completely machine the cylinder surface. In order to achieve the required accuracy and surface quality, you can overlay the movement with a reciprocating stroke. The feed rate of the reciprocating stroke is usually so large that multiple reciprocating strokes per circular path are executed. This is equivalent to grinding with a rapid stroke. The lateral infeeds occur above or below depending on the definition. You can program the feed rate of the reciprocating stroke in the cycle.

Cycle run

1. The control positions the tool above the cylinder based on the **POCKET POSITION Q367**. At **FMAX**, the control then moves the tool to the **CLEARANCE HEIGHT Q260**.
2. At **FMAX**, the tool moves to the starting point in the working plane and then at **F PRE-POSITIONING Q253** to the **SET-UP CLEARANCE Q200**.
3. The grinding tool moves to the starting point in the tool axis. The starting point depends on the **MACHINING DIRECTION Q1031**. If you have defined a reciprocating stroke in **Q1000**, then the control starts the reciprocating stroke.
4. Depending on the parameter **Q1021**, the control laterally infeeds the grinding tool. The control then infeeds in the tool axis.

   Further information: "Infeed", Page 769
5. If the final depth has been reached, then the grinding tool moves for another full circle without a tool axis infeed.
6. The control repeats steps 4 and 5 until the diameter of the finished part **Q223** or the oversize **Q14** has been reached.
7. After the last infeed run, the grinding tool executes the **IDLE RUNS, CONT. END Q457**.
8. The grinding tool leaves the cylinder on a semi-circular path to the safety clearance **Q200** and stops the reciprocating stroke.
9. At **F PRE-POSITIONING Q253**, the control moves the tool to the **SAFETY CLEARANCE Q200** and then at rapid traverse to the **CLEARANCE HEIGHT Q260**.

Infeed

1 The control infeeds the grinding tool in a semi-circle to the **LATERAL INFEED Q534**.
2 The grinding tool executes a full circle and performs any programmed **IDLE RUNS, CONTOUR Q456**.
3 If the area to be traversed in the tool axis is greater than the grinding wheel width **B**, then the cycle moves in a helical path.

Helical path

You can influence the helical path via a pitch in the parameter **Q1032**. The pitch per helical path (= 360°) is relative to the grinding wheel width.

The number of helical paths (= 360°) depends on the pitch and the **DEPTH Q201**. The smaller the pitch, the more helical paths (= 360°) there are.

Example:

- Grinding wheel width **B** = 20 mm
- **Q201 DEPTH** = 50 mm
- **Q1032 PITCH FACTOR** (pitch) = 0.5

The control calculates the relationship between the pitch relative to the grinding wheel width.

Pitch per helical path = $20mm * 0.5 = 10mm$

The control covers the distance of 10 mm in the tool axis within a helix. The **DEPTH Q201** and the pitch per helical path result in five helical paths.

Number of helical paths = $\frac{50mm}{10mm} = 5$

Notes

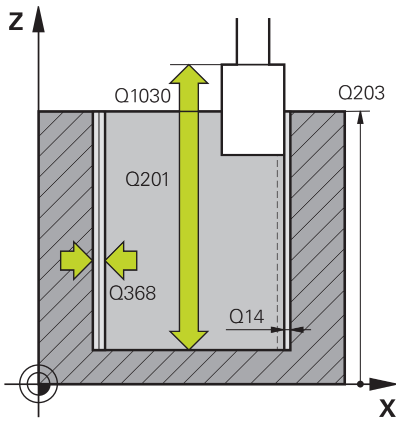
The overrides for the reciprocation movements can be changed by the machine manufacturer.

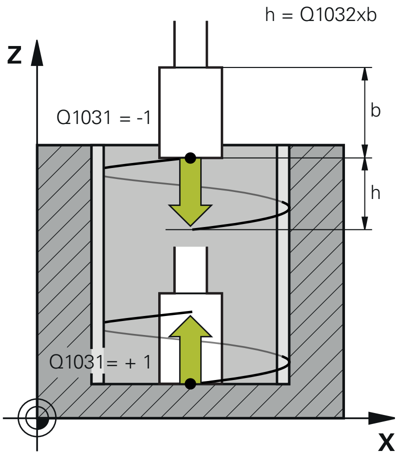
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control always starts the reciprocating stroke in the positive direction.
- The last lateral infeed may be smaller depending on the input.
- The control does not depict the reciprocating movement in the simulation. The reciprocating movement is depicted in the simulation graphics in the **Program run, single block** and **Program run, full sequence** operating modes.
- You can also execute this cycle with a milling cutter. In the case of a milling cutter, the tooth length **LCUTS** equals the width of the grinding wheel.

Notes on programming

- The control assumes that the bottom of the cylinder has a floor. For this reason, you can define an overshoot in **Q1030** only at the surface. If you machine a through hole, for example, then you must take into account the lower overshoot in **DEPTH Q201**.
- If **Q1000=0**, then the control does not execute a superimposed reciprocating movement.

Cycle parameters

Help graphic	Parameter
	Q650 Type of figure? Geometry of the figure: **0**: Pocket **1**: Island Input: **0**, **1**
	Q223 Finished part diameter? Diameter of the fully machined cylinder Input: **0...99999.9999**
	**Q368 Side oversize before machining?** Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+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**
	Q1030 Offset to surface? Position of the upper edge of the tool on the surface. The offset serves as the overshoot path on the surface for the reciprocating stroke. This value has an absolute effect. Input: **0...999.999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**

Help graphic	Parameter
	**Q1031 Machining direction?** Definition of the machining direction. The starting position arises from this. **-1** or **0**: The control machines the contour from up to down during the first infeed cut. **+1**: The control machines the contour from up to down during the first infeed cut. Input: **-1**, **0**, **+1**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q1032 Factor for pitch of helix? You can define the pitch of the helical path (= 360°) with the factor **Q1032**. This results in the infeed depth per helical path (= 360°). **Q1032** is multiplied by the width **B** of the grinding tool. Input: **0.000...1000**
	Q456 Idle runs around contour? Number of times the grinding tool executes the contour without removing material after every infeed. Input: **0...99**
	Q457 Idle runs at contour end? Number of times the grinding tool executes the contour without material removal after the last infeed. Input: **0...99**
	Q1000 Length of reciprocating stroke? Length of the reciprocating movement, parallel to the active tool axis **0**: The control does not perform a reciprocating motion. Input: **0...9999.9999**
	Q1001 Feed rate for reciprocation? Speed of the reciprocating stroke in mm/min Input: **0...999999**
	Q1021 One-sided infeed (0/1)? Position at which the lateral infeed occurs: **0**: Lower and upper lateral infeed **1**: One-sided infeed depending on **Q1031** ■ If **Q1031 = -1**, then the lateral infeed is performed above. ■ If **Q1031 = +1**, then the lateral infeed is performed below. Input: **0**, **1**

Help graphic	Parameter
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q15 Up-cut / climb grinding (-1/+1)? Define the type of contour grinding: **+1**: Climb grinding **-1** or **0**: Up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** 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**

Example

```
11 CYCL DEF 1022 CYLINDER, FAST-STROKE GRINDING ~
    Q650=+0              ;FIGURE TYPE ~
    Q223=+50             ;FINISHED PART DIA. ~
    Q368=+0.1            ;OVERSIZE AT START ~
    Q14=+0               ;ALLOWANCE FOR SIDE ~
    Q367=+0              ;POCKET POSITION ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q1030=+2             ;SURFACE OFFSET ~
    Q201=-20             ;DEPTH ~
    Q1031=-1             ;MACHINING DIRECTION ~
    Q534=+0.05           ;LATERAL INFEED ~
    Q1032=+0.5           ;PITCH FACTOR ~
    Q456=+0              ;IDLE RUNS, CONTOUR ~
    Q457=+0              ;IDLE RUNS, CONT. END ~
    Q1000=+5             ;RECIPROCATING STROKE ~
    Q1001=+5000          ;RECIP. FEED RATE ~
    Q1021=+0             ;ONE-SIDED INFEED ~
    Q207=+50             ;GRINDING FEED RATE ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q15=+1               ;TYPE OF GRINDING ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q200=+2              ;SET-UP CLEARANCE
```

15.13 Cycle 1025 GRINDING CONTOUR (option 156)

ISO programming
G1025

Application

Use Cycle **1025 GRINDING CONTOUR** in combination with Cycle **14 CONTOUR** to grind open and closed contours.

Cycle sequence

1 The control first moves the tool at rapid traverse to the starting position in the X and Y directions and then to clearance height **Q260**.
2 The tool uses rapid traverse to move to set-up clearance **Q200** above the coordinate surface.
3 From there, it moves at the pre-positioning feed rate **Q253** to the depth **Q201**.
4 If programmed, the control performs the approach movement.
5 The cycle starts with the first stepover **Q534**.
6 If programmed, the control performs the number of idle runs **Q456** after each infeed.
7 This process (steps 5 and 6) is repeated until the contour or finishing allowance **Q14** has been reached.
8 After the last infeed, the specified number of air strokes at contour end **Q457** are performed.
9 The control performs the optional departure movement.
10 Finally, the tool is moved at rapid traverse to the clearance height.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The last stepover may be smaller depending on the input.
- Keep in mind that the cycle takes **M109** or **M110** into account, if programmed. In this case, the control will display the feed rate of the center path of the milling tool. The feed rate shown in the status display may thus become lower for inside radii or become higher for outside radii.

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

Note on programming

- If you want to program a reciprocating stroke, you need to define and start it before executing this cycle.

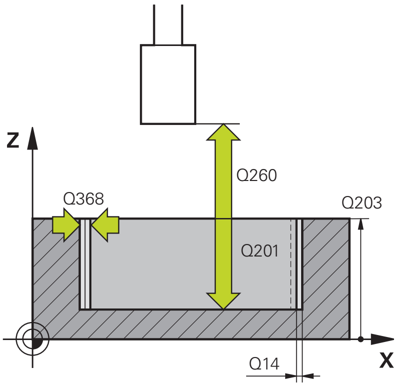
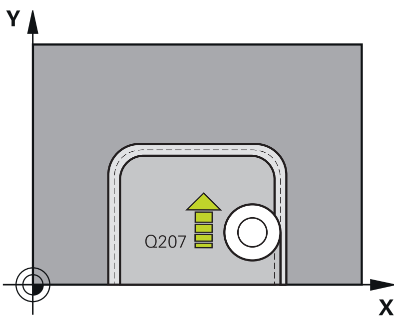
Open contour

- Approach and departure movements for the contour can be programmed using **APPR** and **DEP** or Cycle **270**.

Closed contour

- In the case of a closed contour, only Cycle **270** is available for programming approach and departure movements.
- When grinding a closed contour, it is not possible to alternate between climb and up-cut grinding (**Q15 = 0**). The control issues an error message.
- If you programmed approach and departure movements, the starting position will shift with every infeed. If no approach and departure movements have been programmed, the control automatically generates a vertical movement and the starting position on the contour will not shift.

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**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q368 Side oversize before machining? Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q456 Idle runs around contour? Number of times the grinding tool executes the contour without removing material after every infeed. Input: **0...99**
	**Q457 Idle runs at contour end?** Number of times the grinding tool executes the contour without material removal after the last infeed. Input: **0...99**
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q15 Up-cut / climb grinding (-1/+1)? Define the machining direction of the contours: **+1**: Climb grinding **-1**: Up-cut grinding **0**: Alternating between climb grinding and up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** 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**

Example

```
11 CYCL DEF 1025 GRINDING CONTOUR ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q201=-20            ;DEPTH ~
    Q14=+0              ;ALLOWANCE FOR SIDE ~
    Q368=+0.1           ;OVERSIZE AT START ~
    Q534=+0.05          ;LATERAL INFEED ~
    Q456=+0             ;IDLE RUNS, CONTOUR ~
    Q457=+0             ;IDLE RUNS, CONT. END ~
    Q207=+200           ;GRINDING FEED RATE ~
    Q253=+750           ;F PRE-POSITIONING ~
    Q15=+1              ;TYPE OF GRINDING ~
    Q260=+100           ;CLEARANCE HEIGHT ~
    Q200=+2             ;SET-UP CLEARANCE
```

15.14 Cycle 1030 ACTIVATE WHEEL EDGE (option 156)

ISO programming
G1030

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1030 ACTIVATE WHEEL EDGE** to activate the desired wheel edge. This means that you can change or update the reference point or reference edge. When dressing, you set the workpiece datum to the corresponding wheel edge with this cycle.

For this cycle, a distinction is made between grinding **(FUNCTION MODE MILL / TURN)** and dressing **(FUNCTION DRESS BEGIN / END)**.

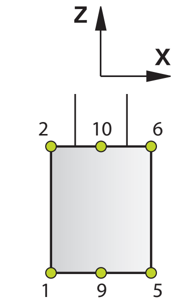
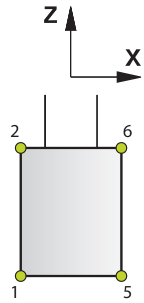
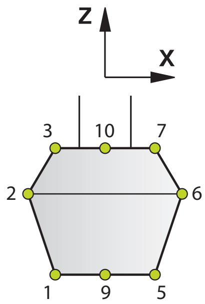
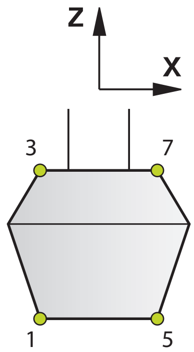
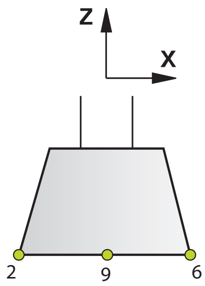
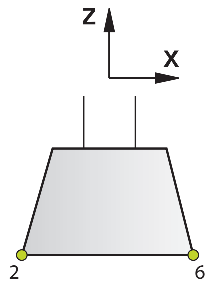
Notes

- This cycle is only permitted in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes if a grinding tool has been activated.
- Cycle **1030** is DEF-active.

Cycle parameters

Help graphic	Parameter
	Q1006 Edge of grinding wheel? Definition of the edge of the grinding tool

Selection of the grinding wheel edges

	Grinding	Dressing
Grinding pin		
Special grinding pin		
Cup wheel		

Example

```
11 CYCL DEF 1030 ACTIVATE WHEEL EDGE ~
    Q1006=+9                ;WHEEL EDGE
```

15.15 Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)

ISO programming
G1032

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1032 GRINDING WHL LENGTH COMPENSATION** to define the overall length of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If initial dressing has not been performed (checkmark for **INIT_D** is not set), then you can change the basic data. Basic data affect both grinding and dressing.

If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

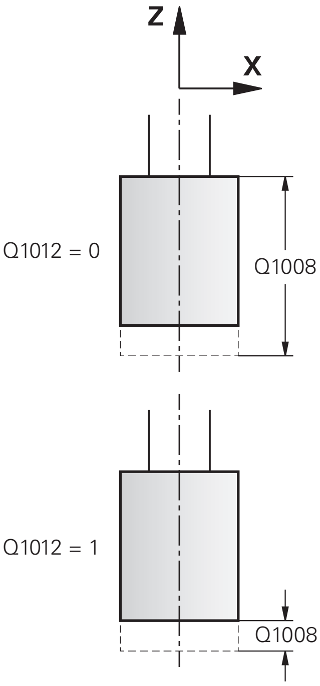
Related topics

- Setting up grinding tools
  Further information: User's Manual for Setup, Testing and Running NC programs

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **1032** is DEF-active.

Cycle parameters

Help graphic	Parameter
	**Q1012 Compens. values (0=abs./1=inc.)?** Definition of the entered length dimension **0**: Entry of the absolute length **1**: Entry of the incremental length Input: **0**, **1**
	Q1008 Comp. value outside edge length? Amount by which the tool is corrected lengthwise based on **Q1012** or by which the tool data are entered without correction. If **Q1012** equals **0**, then the absolute length must be entered. If **Q1012** equals **1**, then the incremental length must be entered. Input: **-999.999...+999.999**
	Q330 Tool number or tool name? Number of name of the grinding tool. Via a softy key, you have the option of applying the tool directly from the tool table. **-1**: The active tool from the tool spindle is used. Input: **-1...99999.9**

Example

```
11 CYCL DEF 1032 GRINDING WHL LENGTH COMPENSATION ~
    Q1012=+1                ;INCR. COMPENSATION ~
    Q1008=+0                ;COMP. OUTSIDE LENGTH ~
    Q330=-1                 ;TOOL
```

15.16 Cycle 1033 GRINDING WHL RADIUS COMPENSATION (option 156)

ISO programming
G1033

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1033 GRINDING WHL RADIUS COMPENSATION** to define the radius of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If initial dressing has not been performed (checkmark for **INIT_D** is not set), then you can change the basic data. Basic data affect both grinding and dressing.

If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Related topics

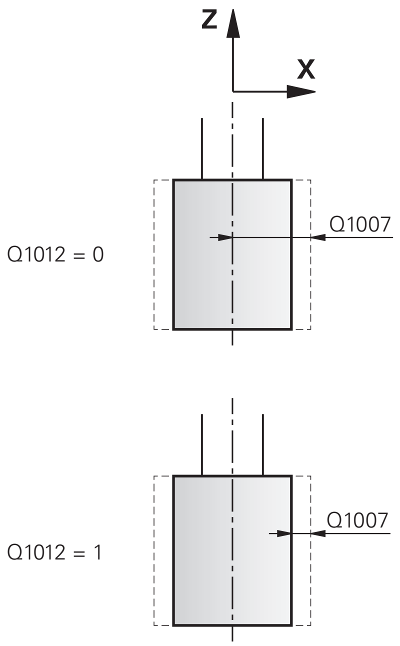
- Setting up grinding tools

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

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **1033** is DEF-active.

Cycle parameters

Help graphic	Parameter
	**Q1012 Compens. values (0=abs./1=inc.)?** Definition of the entered radius dimension **0**: Entry of the absolute radius **1**: Entry of the incremental radius Input: **0**, **1**
	Q1007 Compensation value for radius? Dimension by which the tool radius is compensated for based on **Q1012**. If **Q1012** equals **0**, then the absolute radius must be entered. If **Q1012** equals **1**, then the incremental radius must be entered. Input: **-999.9999...+999.9999**
	Q330 Tool number or tool name? Number of name of the grinding tool. Via a softy key, you have the option of applying the tool directly from the tool table. **-1**: The active tool from the tool spindle is used. Input: **-1...99999.9**

Example

```
11 CYCL DEF 1033 GRINDING WHL RADIUS COMPENSATION ~
    Q1012=+1                ;INCR. COMPENSATION ~
    Q1007=+0                ;RADIUS COMPENSATION ~
    Q330=-1                 ;TOOL
```

15.17 Programming examples

Example of grinding cycles

This programming example illustrates how to machine with a grinding tool.

The NC program uses the following grinding cycles:

- Cycle **1000 DEFINE RECIP. STROKE**
- Cycle **1002 STOP RECIP. STROKE**
- Cycle **1025 GRINDING CONTOUR**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle **1000 DEFINE RECIP. STROKE**
- Define Cycle **14 CONTOUR**
- Define Cycle **1025 GRINDING CONTOUR**
- Define Cycle **1002 STOP RECIP. STROKE**

0 BEGIN PGM GRINDING_CYCLE MM		
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33		
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1		
3 FUNCTION MODE MILL		
4 TOOL CALL 501 Z S20000		; Tool call: grinding tool
5 L Z+30 R0 FMAX M3		
6 CYCL DEF 1000 DEFINE RECIP. STROKE ~		
Q1000=+13	;RECIPROCATING STROKE ~	
Q1001=+25000	;RECIP. FEED RATE ~	
Q1002=+1	;RECIPROCATION TYPE ~	
Q1004=+1	;START RECIP. STROKE	
7 CYCL DEF 14.0 CONTOUR		
8 CYCL DEF 14.1 CONTOUR LABEL1 /2		
9 CYCL DEF 14.2		
10 CYCL DEF 1025 GRINDING CONTOUR ~		
Q203=+0	;SURFACE COORDINATE ~	
Q201=-12	;DEPTH ~	
Q14=+0	;ALLOWANCE FOR SIDE ~	
Q368=+0.2	;OVERSIZE AT START ~	
Q534=+0.05	;LATERAL INFEED ~	
Q456=+2	;IDLE RUNS, CONTOUR ~	
Q457=+3	;IDLE RUNS, CONT. END ~	
Q207=+200	;GRINDING FEED RATE ~	
Q253=+750	;F PRE-POSITIONING ~	
Q15=+1	;TYPE OF GRINDING ~	
Q260=+100	;CLEARANCE HEIGHT ~	
Q200=+2	;SET-UP CLEARANCE	
11 CYCL CALL		; Cycle call: grinding contour

12 L Z+50 R0 FMAX	
13 CYCL DEF 1002 STOP RECIP. STROKE ~	
Q1005=+1 ;CLEAR RECIP. STROKE ~	
Q1010=+0 ;RECIP.STROKE STOPPOS	
14 L Z+250 R0 FMAX	
15 L C+0 R0 FMAX M92	
16 M30	; End of program
17 LBL 1	; Contour subprogram 1
18 L X+3 Y-23 RL	
19 L X-3	
20 CT X-9 Y-16	
21 CT X-7 Y-10	
22 CT X-7 Y+10	
23 CT X-9 Y+16	
24 CT X-3 Y+23	
25 L X+3	
26 CT X+9 Y+16	
27 CT X+7 Y+10	
28 CT X+7 Y-10	
29 CT X+9 Y-16	
30 CT X+3 Y-23	
31 LBL 0	
32 LBL 2	; Contour subprogram 2
33 L X-25 Y-40 RR	
34 L Y+40	
35 L X+25	
36 L Y-40	
37 L X-25	
38 LBL 0	
39 END PGM GRINDING_CYCLE MM	

Example of dressing cycles

This programming example illustrates dressing mode.

The NC program uses the following grinding cycles:

- Cycle **1030 ACTIVATE WHEEL EDGE**
- Cycle **1010 DRESSING DIAMETER**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle **1030 ACTIVATE WHEEL EDGE**
- Tool call: Dressing tool (no mechanical tool change; only a calculated switch-over)
- Cycle **1010 DRESSING DIAMETER**
- Activate **FUNCTION DRESS END**

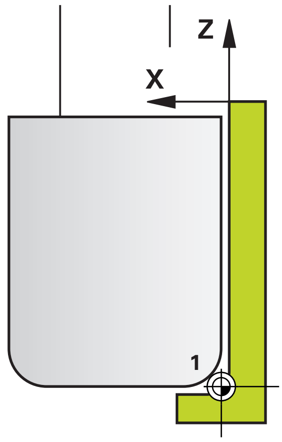
0 BEGIN PGM DRESS_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL 501 Z S20000	; Tool call, grinding wheel
5 M140 MB MAX	
6 L Z+200 R0 FMAX M3	
7 FUNCTION DRESS BEGIN	; Activate dressing procedure
8 CYCL DEF 1030 ACTIVATE WHEEL EDGE ~	
Q1006=+5 ;WHEEL EDGE	
9 TOOL CALL 507	; Tool call, dressing tool
10 L X+5 R0 F2000	
11 L Y+0 R0	
12 L Z-5 M8	
13 CYCL DEF 1010 DRESSING DIAMETER ~	
Q1013=+0 ;DRESSING AMOUNT ~	
Q1018=+300 ;DRESSING FEED RATE ~	
Q1016=+1 ;DRESSING STRATEGY ~	
Q1019=+2 ;NUMBER INFEEDS ~	
Q1020=+3 ;IDLE STROKES ~	
Q1022=+0 ;COUNTER FOR DRESSING ~	
Q330=-1 ;TOOL ~	
Q1011=+0 ;FACTOR VC	
14 FUNCTION DRESS END	; Deactivate dressing procedure
15 M30	; End of program
16 END PGM DRESS_CYCLE MM	

Example of a profile program

Grinding wheel edge no. 1

This example program is for dressing a profile of a grinding wheel. The grinding wheel is curved by the amount of a radius on its outer side.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)



Data to be used:

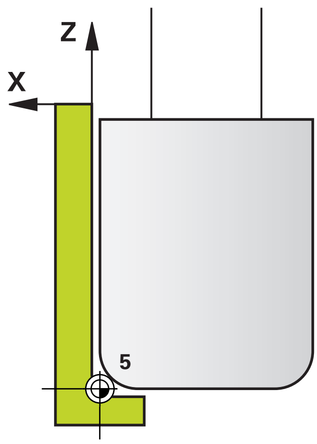
- Grinding wheel edge: 1
- Retraction amount: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm

0 BEGIN PGM 11 MM	
1 L X-5 Z-5 R0 FMAX	; Approach starting position
2 L Z+45 RL FMAX	; Approach starting position
3 L X+0 FQ1018	; Q1018 = Dressing feed rate
4 L Z+0 FQ1018	; Approach radius edge
5 RND R2 FQ1018	; Rounding
6 L X+6 FQ1018	; Approach final position X
7 L Z-5 FQ1018	; Approach final position Z
8 L X-5 Z-5 R0 FMAX	; Approach starting position
9 END PGM 11 MM	

Grinding wheel edge no. 5

This example program is for dressing a profile of a grinding wheel. The grinding wheel is curved by the amount of a radius on its outer side.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)



Data to be used:

- Grinding wheel edge: 5
- Retraction amount: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm

0 BEGIN PGM 12 MM	
1 L X+5 Z-5 R0 FMAX	; Approach starting position
2 L Z+45 RR FMAX	; Approach starting position
3 L X+0 FQ1018	; Q1018 = Dressing feed rate
4 L Z+0 FQ1018	; Approach radius edge
5 RND R2 FQ1018	; Rounding
6 L X-6 FQ1018	; Approach final position X
7 L Z-5 FQ1018	; Approach final position Z
8 L X+5 Z-5 R0 FMAX	; Approach starting position
9 END PGM 11 MM	

16

Tables of Cycles

16.1 Table of cycles

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

ID of User's Manual for Programming of Measuring Cycles for Workpieces and Tools: 1303409-xx

Machining cycles

Cycle number	Cycle name	DEF active	CALL active	Page
7	DATUM SHIFT	■		241
8	MIRRORING	■		244
9	DWELL TIME	■		444
10	ROTATION	■		245
11	SCALING FACTOR	■		247
12	PGM CALL	■		445
13	ORIENTATION	■		447
14	CONTOUR	■		283
18	THREAD CUTTING		■	514
19	WORKING PLANE	■		249
20	CONTOUR DATA	■		287
21	PILOT DRILLING		■	290
22	ROUGH-OUT		■	292
23	FLOOR FINISHING		■	297
24	SIDE FINISHING		■	300
25	CONTOUR TRAIN		■	306
26	AXIS-SPECIFIC SCALING	■		248
27	CYLINDER SURFACE		■	407
28	CYLINDER SURFACE		■	410
29	CYL SURFACE RIDGE		■	415
32	TOLERANCE	■		448
39	CYL. SURFACE CONTOUR		■	419
200	DRILLING		■	78
201	REAMING		■	82
202	BORING		■	84
203	UNIVERSAL DRILLING		■	88
204	BACK BORING		■	94
205	UNIVERSAL PECKING		■	98
206	TAPPING		■	131
207	RIGID TAPPING		■	134

Cycle number	Cycle name	DEF active	CALL active	Page
208	**BORE MILLING**		■	106
209	**TAPPING W/ CHIP BRKG**		■	139
220	**POLAR PATTERN**	■		262
221	**CARTESIAN PATTERN**	■		266
224	**DATAMATRIX CODE PATTERN**	■		270
225	**ENGRAVING**		■	469
232	**FACE MILLING**		■	476
233	**FACE MILLING** (milling direction can be selected, take the side walls into account)		■	225
238	**MEASURE MACHINE STATUS**	■		508
239	**ASCERTAIN THE LOAD**	■		511
240	**CENTERING**		■	122
241	**SINGLE-LIP D.H.DRLNG**		■	111
247	**PRESETTING**	■		255
251	**RECTANGULAR POCKET**		■	179
252	**CIRCULAR POCKET**		■	187
253	**SLOT MILLING**		■	194
254	**CIRCULAR SLOT**		■	201
256	**RECTANGULAR STUD**		■	208
257	**CIRCULAR STUD**		■	214
258	**POLYGON STUD**		■	219
262	**THREAD MILLING**		■	146
263	**THREAD MLLNG/CNTSNKG**		■	151
264	**THREAD DRILLNG/MLLNG**		■	157
265	**HEL. THREAD DRLG/MLG**		■	163
267	**OUTSIDE THREAD MLLNG**		■	168
270	**CONTOUR TRAIN DATA**		■	304
271	**OCM CONTOUR DATA**		■	338
272	**OCM ROUGHING**		■	341
273	**OCM FINISHING FLOOR**		■	356
274	**OCM FINISHING SIDE**		■	360
275	**TROCHOIDAL SLOT**		■	311
276	**THREE-D CONT. TRAIN**		■	317
277	**OCM CHAMFERING**		■	364
285	**DEFINE GEAR**	■		485
286	**GEAR HOBBING**		■	488
287	**GEAR SKIVING**		■	496
291	**COUPLG.TURNG.INTERP.**		■	452

Cycle number	Cycle name	DEF active	CALL active	Page
292	CONTOUR.TURNG.INTRP.		■	459
1271	OCM RECTANGLE	■		371
1272	OCM CIRCLE	■		375
1273	OCM SLOT / RIDGE	■		378
1274	OCM CIRCULAR SLOT	■		382
1278	OCM POLYGON	■		386
1281	OCM RECTANGLE BOUNDARY	■		390
1282	OCM CIRCLE BOUNDARY	■		392

Turning cycles

Cycle number	Cycle name	DEF active	CALL active	Page
800	ADJUST XZ SYSTEM	■		541
801	RESET ROTARY COORDINATE SYSTEM	■		549
810	TURN CONTOUR LONG.		■	583
811	SHOULDER, LONGITDNL.		■	565
812	SHOULDER, LONG. EXT.		■	569
813	TURN PLUNGE CONTOUR LONGITUDINAL		■	574
814	TURN PLUNGE LONGITUDINAL EXT.		■	578
815	CONTOUR-PAR. TURNING		■	588
820	TURN CONTOUR TRANSV.		■	610
821	SHOULDER, FACE		■	592
822	SHOULDER, FACE. EXT.		■	596
823	TURN TRANSVERSE PLUNGE		■	601
824	TURN PLUNGE TRANSVERSE EXT.		■	605
830	THREAD CONTOUR-PARALLEL		■	688
831	THREAD LONGITUDINAL		■	678
832	THREAD EXTENDED		■	683
840	RECESS TURNG, RADIAL		■	633
841	SIMPLE REC. TURNG., RADIAL DIR.		■	615
842	ENH.REC.TURNNG, RAD.		■	619
850	RECESS TURNG, AXIAL		■	638
851	SIMPLE REC TURNG, AX		■	624
852	ENH.REC.TURNING, AX.		■	628
860	CONT. RECESS, RADIAL		■	666
861	SIMPLE RECESS, RADL.		■	643
862	EXPND. RECESS, RADL.		■	648
870	CONT. RECESS, AXIAL		■	672
871	SIMPLE RECESS, AXIAL		■	654
872	EXPND. RECESS, AXIAL		■	659
880	GEAR HOBBING		■	551
882	SIMULTANEOUS ROUGHING FOR TURNING		■	694
883	TURNING SIMULTANEOUS FINISHING		■	701
892	CHECK UNBALANCE	■		560

Grinding cycles

Cycle number	Cycle name	DEF active	CALL active	Page
1000	DEFINE RECIP. STROKE	■		722
1001	START RECIP. STROKE	■		725
1002	STOP RECIP. STROKE	■		726
1010	DRESSING DIAMETER	■		731
1015	PROFILE DRESSING	■		736
1016	DRESSING OF CUP WHEEL	■		742
1017	DRESSING WITH DRESSING ROLL	■		747
1018	RECESSING WITH DRESSING ROLL	■		754
1021	CYLINDER, SLOW-STROKE GRINDING		■	760
1022	CYLINDER, FAST-STROKE GRINDING		■	768
1025	GRINDING CONTOUR		■	774
1030	ACTIVATE WHEEL EDGE	■		778
1032	GRINDING WHL LENGTH COMPENSATION	■		780
1033	GRINDING WHL RADIUS COMPENSATION	■		782

Index

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

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
1303406-23 · Ver03 · SW18 · 10/2023 · H · Printed in Germany