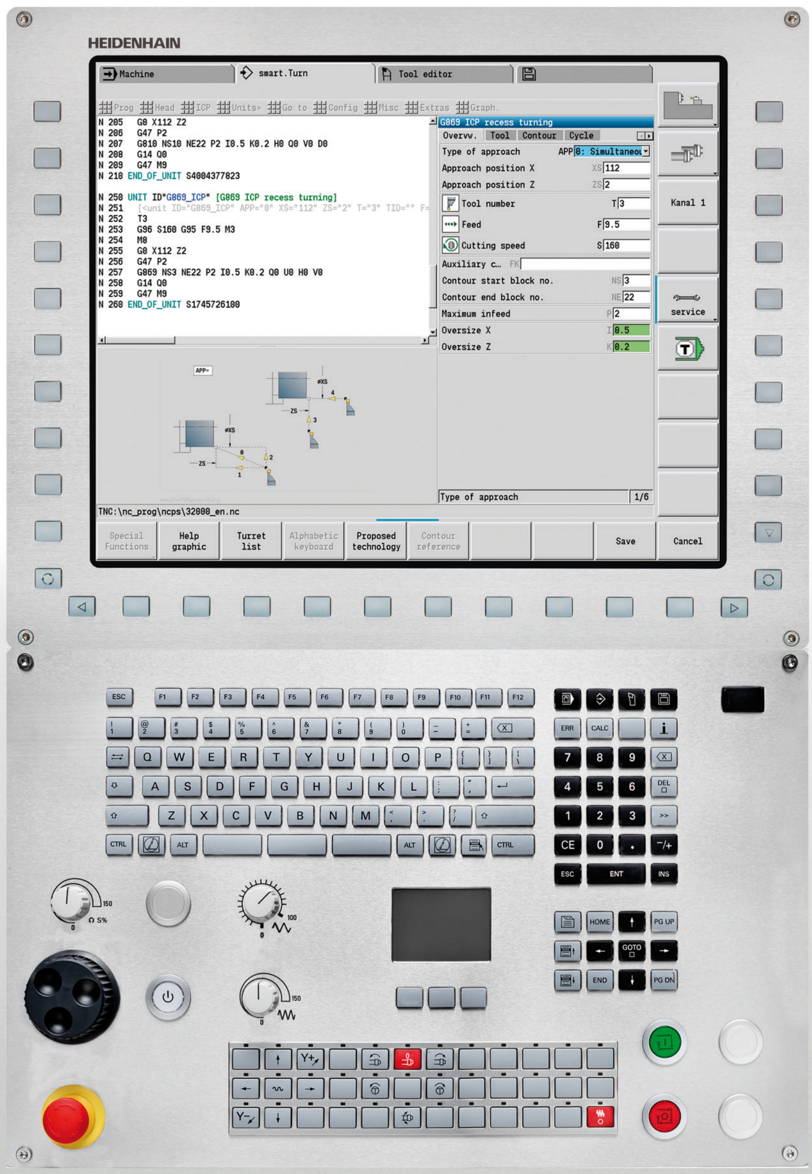
HEIDENHAIN

User's Manual



MANUALplus 620
CNC PILOT 620/640
smart.Turn and DIN Programming

NC Software
548328-05
548430-01
548431-01
688945-03
688946-01
688947-01

English (en)
8/2013

smart.Turn and DIN PLUS programming

This manual describes functions and features provided by lathe controls as of the following NC software numbers.

Control	NC software number
MANUALplus 620E	548328-05
MANUALplus 620 (HEROS 5)	548430-01
MANUALplus 620E (HEROS 5)	548431-01
CNC PILOT 620E	688945-03
CNC PILOT 640 (HEROS 5)	688946-01
CNC PILOT 640E (HEROS 5)	688947-01

The suffix **E** indicates the export version of the control. The export version of the control has the following limitations:

- Simultaneous linear movement in up to 4 axes

HEROS 5 identifies the new operating system of HSCI-based controls.

Machine operation and cycle programming are described in the MANUALplus 620 (ID 634864-xx) and CNC PILOT 620/640 (ID 730870-xx) User's Manuals. Please contact HEIDENHAIN if you require a copy of one of these manuals.

The machine manufacturer adapts the features offered by the control to the capabilities of the specific machine tool by setting machine parameters. Therefore, some of the functions described in this manual may not be among the features provided by the Control on your machine tool.

Some of the Control functions which are not available on every machine are:

- Positioning of spindle (M19) and driven tool
- Operations with the C or Y axis

Please contact your machine manufacturer for detailed information on the features that are supported by your machine tool.

Many machine manufacturers and HEIDENHAIN offer programming courses. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other Control users.

HEIDENHAIN also offers the DataPilot programming station for PCs, which is designed for use with the respective control. The DataPilot is excellently suited for both shop-floor programming as well as off-location program creation and production planning. It is also ideal for training purposes. The DataPilot can be run on PCs with WINDOWS operating systems.

Control	Programming station	NC Software
MANUALplus 620	DataPilot MP620	634132-05
CNC PILOT 620	DataPilot CP620	729665-03
CNC PILOT 640	DataPilot CP640	729666-01

Intended place of operation

The MANUALplus 620, CNC PILOT 620/640 complies with the limits for a Class A devices in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

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

- Organization mode of operation
- Second soft-key row
- LICENSE INFO soft key

New functions of software 548328-03

- The simulation function now also provides a 3-D view of the workpiece blank and finished part. Workpieces can be depicted as a solid-body model or transparent. The graphic can be both rotated around the main axes and displayed in a three-quarter section view (see the User's Manual).
- Simplified contour editing with ICP: When graphically supported interactive contour description with ICP is active, contour-element editing can now be selected directly by soft key (see the User's Manual).
- Recessing with cut segmentation: Cut segmentation for recessing depth is now available for the contour recessing cycle G860. (see page 71)
- The parameters XA and ZA (starting point of blank) were added to the roughing cycles G810, G820, G830 and G835. This makes it possible to start the cutting process at any desired diameter of the contour by entering a starting point. (see page 60)
- Workpiece measurement: The Control now supports workpiece measurement with touch probes. A sample cycle for measurement of workpieces is available in the control. Additionally, the machine manufacturers can offer specially developed measuring cycles that are individually adapted to the machine series (see the User's Manual).
- A speed limit that is only effective for the respective cycle can now be defined for the parting cycle in Teach-in mode and smart.Turn (see page 75)
- In the roughing and drilling cycles, an intermittent feed rate can now be entered for chip breaking. (see page 60)
- The approach angle and departure angle have been added to the ICP cutting cycles. (see page 65)
- The graphic window can now be activated automatically when the cursor is located in the contour description. (see page 44)

New functions of software 688945-02 and 548328-04

- In the program simulation, the current contour description (of workpiece blank and finished part) can be mirrored and saved. In smart.Turn, these contours can be reinserted (see User's Manual)
- On machines with counterspindle, the workpiece spindle can now be selected in the TSF menu (see User’s Manual)
- On machines with a counterspindle, it’s datum can be shifted (see User’s Manual)
- The user documentation is now also in the context-sensitive help system TURNguide (see User’s Manual)
- You can make your own project folder in the project management, so that you can centrally manage associated files (see User’s Manual)
- With a manual tool change system it is possible to insert tools that are not in the turret during a program run (see User’s Manual)
- Engraving cycles are now available in the Teach-In mode of operation (see User’s Manual)
- During tool data backup, you can now select in a dialog window the data to be saved or restored (see User’s Manual)
- The G30 function is now available for converting G functions, M functions or spindle numbers, as well as for mirroring traverse paths and tool dimensions (see "Converting and mirroring G30" on page 378)
- The "traverse to a fixed stop" function (G916) is now available for transferring the workpiece to the second traversable spindle or for pressing the tailstock against the workpiece.(see "Traversing to a fixed stop G916" on page 382)
- The G925 function makes it possible to define and monitor the maximum contact force for an axis. This function can be applied to use the opposing spindle as a mechatronic tailstock, for example (see "Force reduction G925" on page 385)
- Controlled parting using servo-lag monitoring (G917) can now be activated to prevent collisions caused by incomplete parting processes.(see "Controlled parting using lag error monitoring G917" on page 384)

- The spindle synchronization option G720 synchronizes the shaft speeds of two or more spindles so that they rotate synchronously with a gear ratio or a defined offset.(see "Spindle synchronization G720" on page 380)
- In combination with the synchronization (G720) of main spindle and tool spindle, the new "Hobbing" cycle (G808) is available for milling external teeth and profiles.(see "Hobbing G808" on page 520)
- With G924, a "fluctuating speed" can now be programmed to prevent resonance (see "Fluctuating spindle speed G924" on page 374)

New functions of software 548328-05, 54843x-01, 688945-03 ad 688946-01

- On machines with a B axis it is now also possible to drill, bore, and mill in oblique planes. In addition to this, the B axis enables you to use tools even more flexibly during turning (see "Tilted working plane" on page 558).
- The control now provides numerous touch probe cycles for various applications (see "General information on touch probe cycles (software option)" on page 430):
  - Calibrating a touch trigger probe
  - Measuring circles, circle segments, angle and position of the C axis
  - Misalignment compensation
  - Single- point and double-point measurement
  - Finding a hole or stud
  - Zero point setting in the Z or C axis
  - Automatic tool measurement
- The new TURN PLUS function automatically generates NC programs for turning and milling operations based on a fixed machining sequence (see "TURN PLUS mode of operation" on page 530).
- The G940 function now provides a way to calculate the tool lengths in the basic (definition) position of the B axis (see "Calculate variables automatically G940" on page 375)
- For machining operations that require rechucking, you can define a separation point on the contour description with G44 (see "Separation point G44" on page 215).
- The G927 function enables you to convert tool lengths to the reference position of the tool (B axis = 0) (see "Convert lengths G927" on page 375).
- Recesses that were defined with G22 can now be machined with the new Cycle 870 ICP Recessing (see ""ICP recessing" unit" on page 77).

About this manual

The symbols used in this manual are described below.

This symbol indicates that important information about the function described must be considered.

This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator

This symbol indicates that the described function must be adapted by the machine tool builder. The function described may therefore vary depending on the machine.

This symbol indicates that you can find detailed information about a function in another manual.

Do you want any changes, or have you found any errors?

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **tnc-userdoc@heidenhain.de**.

Contents

3 smart.Turn units for the Y axis 161

5 Touch probe cycles 429

1

NC programming

1.1 smart.Turn and DIN (ISO) programming

The Control supports the following types of NC programming:

- **Conventional DIN/ISO programming:** You program the basic contour with line segments, circular arcs and simple turning cycles. Use the smart.Turn editor in ISO mode.
- **"DIN PLUS" (ISO) programming:** The geometrical description of the workpiece and the machining process are separated. You first program the geometry of the blank and finished part. Then you machine the workpiece, using contour-related turning cycles. Use the smart.Turn editor in ISO mode.
- **smart.Turn programming:** The geometrical description of the workpiece and the machining process are separated. You program the geometry of the blank and finished part, and you program the machining blocks as units. Use the smart.Turn editor in unit mode.

Depending on the type and complexity of your machining task, you can use either simple DIN/ISO programming, "DIN PLUS" (ISO) programming or smart.Turn programming. All three named programming modes can be combined in one NC program.

In DIN PLUS and smart.Turn programming, contours can be described with ICP interactive graphics. ICP saves the contour descriptions as G codes in the NC program.

Parallel operation: While you are editing and testing programs, your machine can execute **another** NC program.

Contour follow-up

The Control uses the **contour follow-up** function in DIN PLUS and smart.Turn programs. The Control takes the blank part as a basis and accounts for each cut and each cycle when regenerating the contour. Thus you can inspect the current contour of the workpiece during each machining stage. With the "contour follow-up" function, the Control optimizes the paths for approach and departure and avoids air cuts.

Contour regeneration is only available for turning operations when a blank part has been programmed. It also works with auxiliary contours.

Structured NC program

smart.Turn and DIN PLUS programs are structured in fixed sections. The following program sections are created automatically in a new NC program:

- **Program head:** Contains information on the material of the workpiece, the unit of measure as well as further organizational data and setup information as a comment.
- **Chucking equipment:** Description of the workpiece clamping situation.
- **Blank:** The workpiece blank is stored. Programming a blank activates the contour follow-up.
- **Finished part:** The finished part is stored. It is advisable to describe the complete workpiece as a finished part. The units or fixed cycles use NS and NE to indicate the workpiece section to be machined.
- **Machining:** Use units or cycles to program the individual machining steps. In a smart.Turn program, the START unit is located at the beginning of the machining process, and the END unit at the end.
- **End:** Indicates the end of the NC program.

If required, for example for machining with the C axis or when programming with variables, you add further program sections.

Use ICP (Interactive Contour Programming) for describing blank and finished parts.

Example: "Structured smart.Turn program"

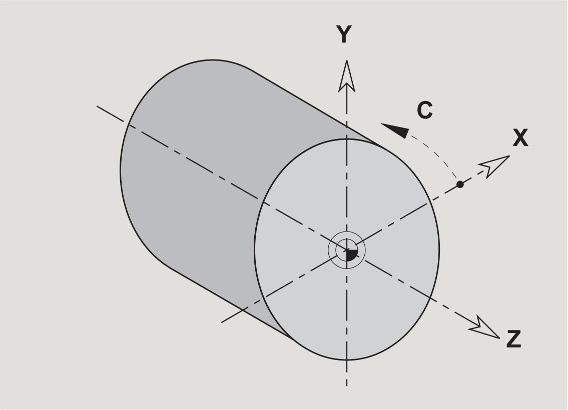
```
HEADER
#UNIT               METRIC
#MATERIAL           Steel
#MACHINE            Automatic lathe
#DRAWING            356_787.9
#CLAMP. PRESS.      20
#COMPANY            Turn & Co

TURRET
T1    ID"038_111_01"
T2    ID"006_151_A"

CHUCKING EQUIPMENT 1
H0 D0 Z200 B20 O-100 X120 K12 Q4

BLANK
N1 G20 X120 Z120 K2

FINISHED PART
N2 G0 X0 Z0
N3 G1 X20 BR3
N4 G1 Z-24
. . .
MACHINING
N50 UNIT ID"START" [Program beginning]
N52 G26 S4000
N53 G59 Z320
N54 G14 Q0
N25 END_OF_UNIT
. . .
[Machining commands]
. . .
N9900 UNIT ID"END" [End of program]
N9902 M30
N9903 END_OF_UNIT
END
```

Linear and rotary axes

Principal axes: Coordinates of the X, Y and Z axes refer to the workpiece datum.

C axis as reference axis:

- Angle data are with given respect to the zero point of the C axis.
- C-axis contours and C-axis operations:
  - Positions on the front/rear face are entered in Cartesian coordinates (XK, YK), or polar coordinates (X, C)
  - Positions on the lateral surface are entered in polar coordinates (Z, C). Instead of C, the **linear value CY** is used ("unrolled" reference diameter).



- The smart.Turn editor respects only address letters of the configured axes.

Units of measure

You write NC programs in metric or inch values. The unit of measure is defined in the "Unit" box (See "HEADER section" on page 47.).

Once the unit of measure has been defined, it cannot be edited any longer.

Elements of an NC program

An NC program consists of the following elements:

- Program name
- Program section codes
- Units
- NC blocks
- Commands for program structuring
- Comment blocks

The **program name** begins with "%" followed by up to 40 characters (numbers, uppercase letters or underscore; no diacritical marks) and the extension "nc" for main programs or "ncs" for subprograms. The first character must be a number or a letter.

Program section codes: When you create a new NC program, certain program section codes are already entered. You can add new codes or delete existing ones, depending on your program requirements. An NC program must contain at least the MACHINING and END section codes.

The **unit** begins with this keyword followed by the identification of the unit (ID"G..."). The following lines contain the G, M and T functions of this machining block. The unit ends with END_OF_UNIT followed by a check digit.

NC blocks begin with an N followed by a block number (with up to five digits). The block numbers do not affect the sequence in which the program blocks are executed. They are only intended for identifying the individual NC blocks.

The NC blocks of the HEADER and TURRET sections are not included in the block number organization of the editor.

Program branches, **program repeats** and **subprograms** can be used to structure the program (example: machining the front/back of a bar, etc.).

Input and output: With "input" you can influence the flow of the NC program. Using "output," you can communicate with the machinist. Example: The machinist is required to check measuring points and update compensation values.

Comments are enclosed in brackets "[...]." They are located at the end of an NC block or in a separate NC block. Press the key combination **CTRL+K** to convert an existing block into a comment (and vice versa).

You can also enclose more than one program line in square brackets to mark them as a comment. To do this, enter a comment containing the character "[" and conclude the section by entering another comment containing the character "]".

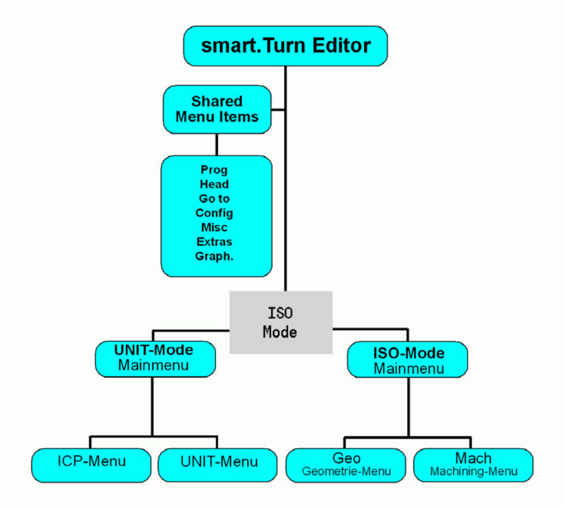
1.2 The smart.Turn editor

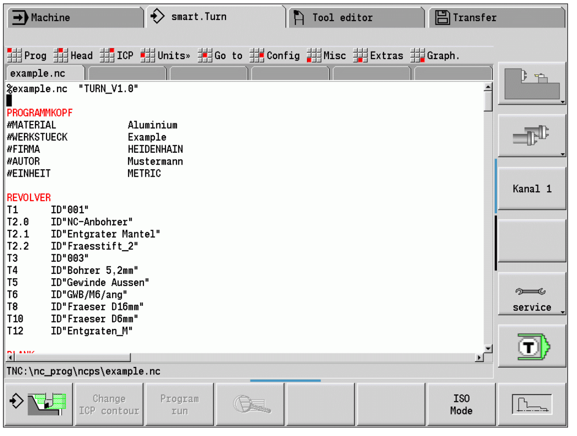
Menu structure

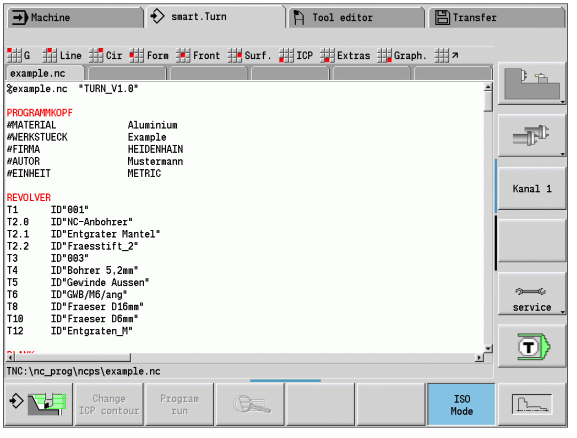
You can select the following editor modes in the smart.Turn editor:

- Unit programming (standard)
- DIN/ISO mode (DIN PLUS and DIN 66025)

The menu structure of the smart.Turn editor is shown in the illustration at right. Many menu items are used in both modes. The menus differ in the area of geometry and part programming. In DIN/ISO mode the menu items "Geo(metry)" and "Mach(ining)" are displayed instead of the menu items "ICP" and "Units" (see illustrations at lower right). You can switch between the editor modes by soft key.







ISO Mode	▶ Switches between the Unit mode and DIN/ISO mode

For special cases you can change to the text-editor mode in order to edit character-by-character without syntax checking. The setting is made in the Configuration / Input mode menu item.

For a description of the functions, please refer to the following chapters:

- Shared menu items: See "Menu structure" on page 38.
- ICP functions: Chapter 5 in the User's Manual
- Units for turning and C-axis machining: See "smart.Turn units" on page 57.
- Units for Y-axis machining: See "smart.Turn units for the Y axis" on page 161.
- G codes for turning and C-axis machining (geometry and machining): See "DIN programming" on page 183.
- G codes for Y-axis machining (geometry and machining): See "DIN programming for the Y axis" on page 477.

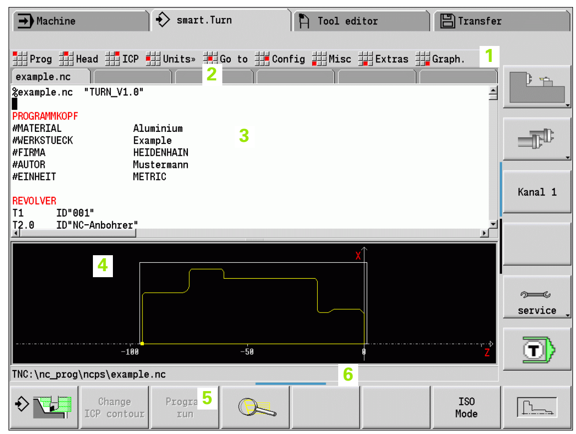
Parallel editing

Up to 6 NC programs can be opened simultaneously in the smart.Turn editor. The editor shows the names of the open programs in the tab bar. If you have changed the NC program, the editor displays the name in red.

You can program in the smart.Turn editor while the machine is running a program in the automatic mode.

- The smart.Turn editor saves all open programs with every mode change.
- The program running in the automatic mode cannot be edited.

Screen layout



1 Menu bar
2 NC program bar with the name of the loaded NC programs. The selected program is marked.
3 Program window
4 Contour display or large program window
5 Soft keys
6 Status bar

Selecting the editor functions

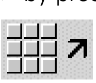
The functions of the smart.Turn editor are contained in the main menu and various submenus.

The submenus can be called by:

- selecting the desired menu item
- positioning the cursor in the respective program section

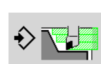
You can access the higher-level menu:

- by pressing the ESC key



- by using the menu item

Soft keys: Soft keys are available for fast switching to "neighboring operating modes," for changing the editing window and for activating the graphics.

Soft keys with active program window	
	Starts the current program in the simulation.
Change ICP contour	Opens the contour, in which the cursor is located, in ICP.
	Activates the zoom function in the contour display.
ISO Mode	Switches between the Unit mode and DIN/ISO mode.
	Activates the contour display and starts redrawing the contour.

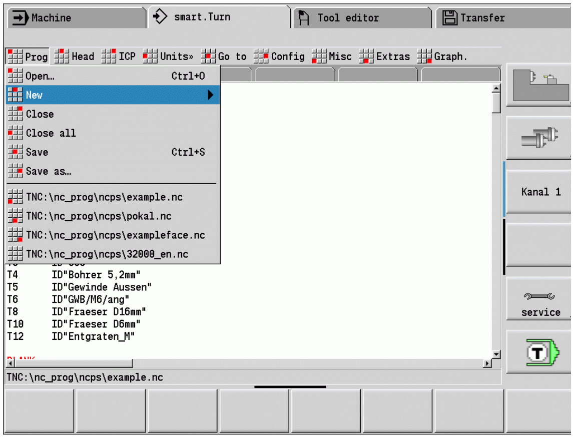
Shared menu items

The menu items described below are used both in smart.Turn mode and in DIN/ISO mode.

"Program management" pull-down menu

The **"Prog" pull-down menu** (program management) contains the following functions for NC main and subprograms:

- **Open**: Load existing programs
- **New**: Create new programs
- **Close**: The selected program is closed
- **Close All**: All open programs are closed
- **Save**: The selected program is saved
- **Save As**: The selected program is saved under a new name
- Direct opening of the last four programs



When an NC program is opened or when a new NC program is created, the soft-key row is switched to the **sorting and organization functions**. See "Sorting, file organization" on page 45..

"Head" pull-down menu (program head)

The **"Head" pull-down menu** (program head) contains functions for editing the program head and the tool list.

- **Program head**: Edit the program head
- **Go to chucking equipment**: Positions the cursor in the "chucking equipment" section
- **Insert chucking equipment:** Describe how the workpiece is clamped
- **Go to tool list**: Positions the cursor in the TURRET section
- **Set up the tool list**: Activates the "Set up tool list" function (see page 53)

"ICP" pull-down menu

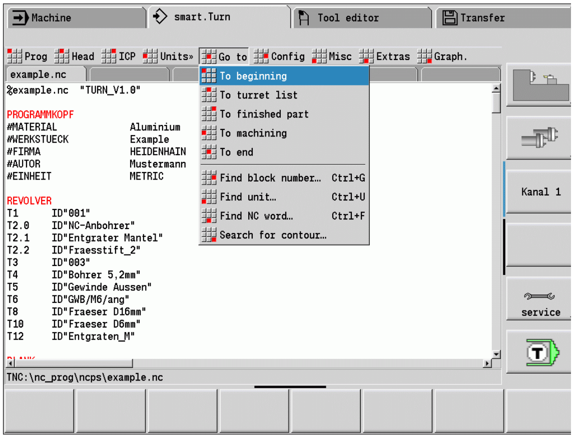
The **"ICP" pull-down menu** (Interactive Contour Programming) contains the following functions:

- **Contour editing**: Change the current contour (cursor position)
- **Workpiece blank**: Edit the description of the workpiece blank
- **Finished part**: Edit the description of the finished part
- **New auxiliary blank**: Create a new auxiliary workpiece blank
- **New aux. contour**: Create a new auxiliary contour
- **C axis ...**: Create patterns and milling contours on the front face and lateral surface
- **Y axis ...**: Create patterns and milling contours in the XY and YZ planes

"Goto" pull-down menu

The **"Goto" pull-down menu** contains the following jump and search functions:

- Jump targets—The editor positions the cursor to the selected jump target:
  - **To beginning**
  - **To tool table**
  - **To finished part**
  - **To machining**
  - **To end**
- Search functions
  - **Find block number**: You specify a certain block number. The editor jumps to this block number if it exists.
  - **Find unit**: The editor opens the list of units available in the program. Select the desired unit.
  - **Find NC word**: The editor opens the dialog for entering the desired NC word. You can use the soft keys to search forward or backward.
  - **Search for contour**: The editor opens the list of contours available in the program. Select the desired contour.



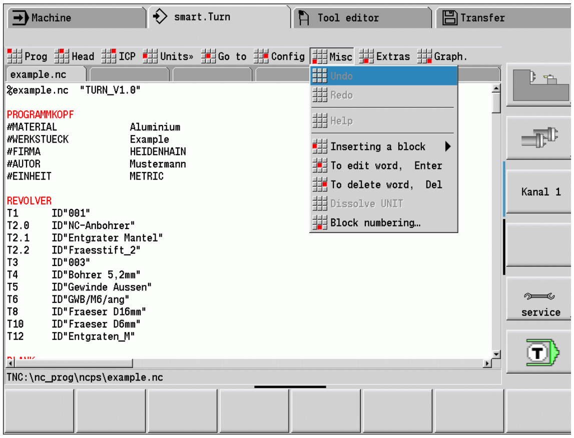
"Configuration" pull-down menu

The **"Config" pull-down menu** (Configuration) contains the following functions:

- **Input mode ...**: Define the input mode
  - **... NC editor (word-by-word)**: The editor works in the NC mode (word by word)
  - **... Text editor (character)**: The editor works character by character (no syntax checking)
- **Settings ...**
  - **... Save**: The editor memorizes the open NC programs and the respective cursor positions.
  - **... Load last saved** setting: Restores the last saved condition of the editor.
- **Technology data**: Starts the technology editor

"Miscellaneous" pull-down menu

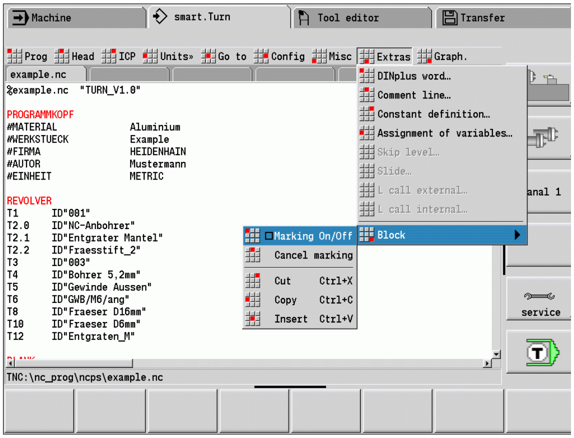
The **"Misc" pull-down menu** (Miscellaneous) contains the following functions:



- **Insert block ...**
  - **... W/o block no.**: The editor inserts an empty line at the cursor position (without block number).
  - **... With block no.**: The editor inserts an empty line at the cursor position (with block number). **Alternative:** When you press the INS key, the editor inserts a block with block number.
  - **... Comment at line end**: The editor inserts a comment at the end of the line in which the cursor is located.
- **Edit word**: You can edit the NC word at which the cursor is located.
- **Delete word**: The editor deletes the NC parameter at the cursor position.
- **Dissolve unit**: Position the cursor to the first line of a unit before selecting this menu item. The editor cancels the brackets around the unit. The unit dialog can no longer be used for this machining block, but you can edit the machining block as desired.
- **Block numbering**: The block numbering settings are the starting block number and the block-number increment. The first NC block receives the starting block number and the block-number increment is added for each further NC block. The settings for starting block number and block number increment are tied with the NC program.

"Extras" pull-down menu

The **"Extras" pull-down menu** contains the following functions:



- **DIN PLUS word**: The editor opens the selection list with all DIN PLUS words in alphabetical order. Select the desired instruction for program structuring or the input/output command. The editor inserts the DIN PLUS word at the cursor position.
- **Comment line**: The comment is inserted above the position of the cursor.
- **Constant definition**: The expression is inserted above the position of the cursor. If the DIN PLUS word "CONST" is not present yet, it is also inserted.
- **Assignment of variables**: Inserts a variable instruction.
- **L call external** (the subprogram is in a separate file): The editor opens the file selection window for subprograms. Select the subprogram and fill out the subprogram dialog. The control searches for subprograms in the sequence: current project, standard directory and then machine manufacturer directory.
- **L call internal** (the subprogram is contained in the main program): The editor opens the subprogram dialog.
- **Block** functions. This pull-down menu contains functions for marking, copying and deleting sections.
  - **Marking On/Off**: Activates/Deactivates the marking mode during cursor movement.
  - **Cancel marking**: After calling the menu item, no part of the program is marked.
  - **Cut**: Deletes the marked part of the program and copies it to the clipboard.
  - **Copy**: Copies the marked part of the program into the clipboard.
  - **Insert**: Inserts the contents of the clipboard at the cursor position. Any parts of the program that are marked are replaced by the contents of the clipboard.

"Graphics" pull-down menu

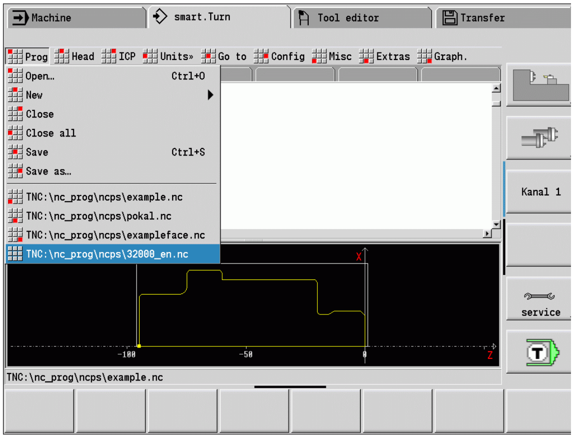
The **"Graph." pull-down menu** contains the following functions (see figure at right):

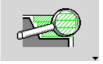
- **Graphic On**: Activates the graphic window or updates the displayed contour. As an alternative, you can use the soft key (see table at right).
- **Graphic Off**: Closes the graphic window.
- **Graphic for Automatic**: The graphic window is activated when the cursor is located in the contour description.
- **Window**: Sets the graphic window. During editing, the Control displays programmed contours in up to four graphic windows. Set the desired windows.
- **Magnifier on**: Activates the zoom function. As an alternative, you can use the soft key (see table at right).

The graphic window:

- Colors in contour graphics:
  - White: workpiece blank and auxiliary blank
  - Yellow: finished part
  - Blue: auxiliary contours
  - Red: contour element at the current cursor position. The arrow point indicates the direction of machining.
- When programming fixed cycles, you can use the displayed contour for establishing block references.
- Using the zoom functions, you can magnify, reduce or shift details.

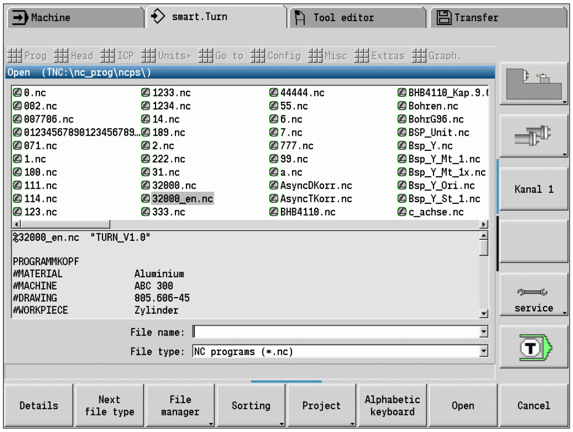
> - Additions/changes to the contour will not be considered until the GRAPHICS soft key is pressed again.
> - Unambiguous NC block numbers are a prerequisite for the contour display!



Soft keys with active program window	
	Activates the contour display and starts redrawing the contour.
	Opens the soft-key menu for the zoom functions and displays the zoom frame.

Sorting, file organization

When an NC program is opened or when a new NC program is created, the soft-key row is switched to the sorting and organization functions. Use the soft keys to select the order in which the programs are to be displayed, or use the functions for copying, deleting, etc.



Soft keys file manager

Delete	Deletes the selected program after confirmation prompt
Rename	Makes it possible to change the program name
Copy	Copies the selected program
Write protection	Switches the write protection attribute on or off for the selected program
Alphabetic keyboard	Activates the alphabetic keyboard

Soft keys for sorting

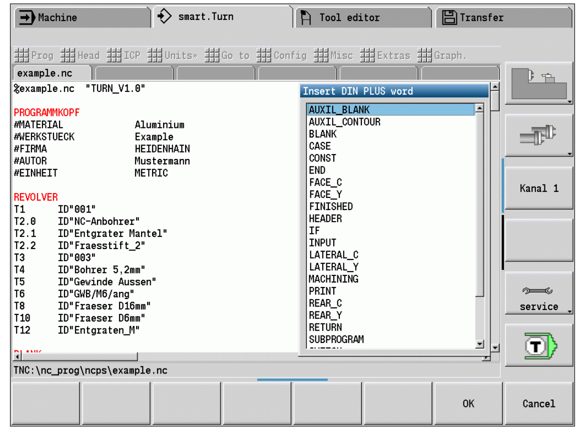
Details	Display of file attributes: size, date, time
Sort by file name	Sort by file name
Sort by size	Sort by file size
Sort by date	Sort by creation date or change date
Reverse sorting	Reversal of the sorting direction
Open	Opens the selected program

1.3 Program section code

A new NC program is already provided with section codes. You can add new codes or delete existing ones, depending on your program requirements. An NC program must contain at least the MACHINING and END section codes.

Further program section codes are available in the "Insert DIN PLUS word" selection list ("Extras > DIN PLUS word" menu item). The Control enters the program section code at the correct position or at the current position.

German program section codes are used when German is set as the conversational language. All other languages use English program section codes.



Overview of program section codes

German	English	
Program head		
PROGRAMMKOPF	HEADER	Page 47
SPANNMITTEL	CHUCKING EQUIPMENT (CLAMPS)	Page 48
REVOLVER	TURRET	Page 48
Contour definition		
ROHTEIL	BLANK	Page 49
FERTIGTEIL	FINISHED	Page 49
HILFSKONTUR	AUXIL_CONTOUR	Page 49
HILFSROHTEIL	AUXIL_BLANK	Page 49
C-axis contours		
STIRN	FACE_C	Page 49
RUECKSEITE	REAR_C	Page 49
MANTEL	LATERAL_C	Page 49
Y-axis contours		
STIRN_Y	FACE_Y	Page 49
RUECKSEITE_Y	REAR_Y	Page 49
MANTEL_Y	LATERAL_Y	Page 50
Workpiece machining		
BEARBEITUNG	MACHINING	Page 51
ENDE	END	Page 51

Example: Program section codes

```
. . .  [Sections of the contour description]
BLANK
N1 G20 X100 Z220 K1

FINISHED PART
N2 G0 X60 Z0
N3 G1 Z-70
. . .
FRONT Z-25
N31 G308 ID"01" P-10
N32 G402 Q5 K110 A0 Wi72 V2 XK0 YK0
N33 G300 B5 P10 W118 A0
N34 G309

FRONT Z0
N35 G308  ID"02" P-6
N36 G307 XK0 YK0 Q6 A0 K34.641
N37 G309
. . .
```

Overview of program section codes		
German	**English**	
Subprograms		
UNTERPROGRAMM	**SUBPROGRAM**	Page 51
RETURN	**RETURN**	Page 51
Others		
CONST	**CONST**	Page 52
VAR	**VAR**	Page 52

For more than one independent contour definition for drilling/milling, use the program section codes (FRONT, SURFACE, etc.) each time.

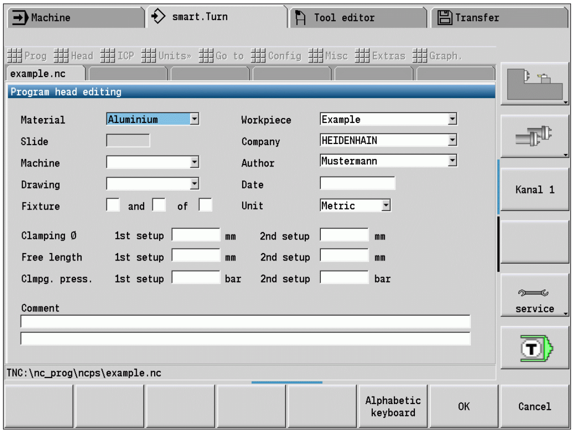
HEADER section

Instructions and information in the program head (HEADER):

- **Unit:**
  - Select dimensional system in millimeters or inches
  - No entry: The unit set in the user parameter is used.
- The other fields contain **organizational information** and **set-up information**, which do not influence the machining process.

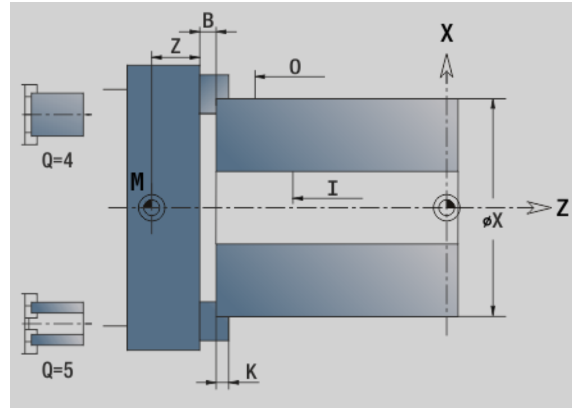
Information contained in the program head is preceded by "#" in the NC program.

You can only select a unit when creating a new NC program. It is not possible to post-edit this entry.



CHUCKING EQUIPMENT section

In the CHUCKING EQUIPMENT program section you describe how the workpiece is clamped.



Parameters

H	No. of clamping (no. of chuck) (always program H=0)
D	Spindle number AWG
Z	Edge of chuck
B	Width of chuck
O	Cutting limitation, outside
I	Cutting limitation, inside
X	Clamping diameter of workpiece blank
K	Clamp length
Q	Chuck form ■ 4: Externally clamped ■ 5: Internally clamped
V	Shaft machining AWG ■ 0: Chuck: Automatic separation points at largest and smallest diameter ■ 1: Shaft/chuck: Machining also starting from the chuck ■ 2: Shaft/face driver: Outside contour can be machined completely

TURRET section

The TURRET program section defines the assignment of the tool carrier. For every assigned turret pocket, the **tool ID number** is entered. For multipoint tools, every cutting edge is entered in the turret list.

- If you **do not program the TURRET,** the tools entered in the tool list of the Machine operating mode will be used.

Example: Turret table

```
. . .
TURRET
T1 ID"342-300.1"
T2 ID"C44003"
. . .
```

BLANK section

In this program section, you describe the contour of the workpiece blank.

AUXIL_BLANK section

In the AUXIL_ BLANK section, you define additional workpiece blanks, which can be activated with G702 when required.

FINISHED section

In this program section, you describe the contour of the finished part. After the **FINISHED** section you use additional section codes such as FACE, LATERAL, etc.

AUXIL_CONTOUR section

In this program section, you describe the auxiliary turning contours.

FACE, REAR sections

In this program section you describe the front and rear side contours to be machined with the C axis. The program section defines the position of the contour in Z direction.

Parameter

Z Position of the front/rear-face contour

LATERAL section

In this program section you describe the lateral surface contours to be machined with the C axis. The program section defines the position of the contour in X direction.

Parameter

X Reference diameter of lateral-surface contours.

FRONT_Y, REAR_Y sections

For lathes with Y axis, these program section codes define the XY plane (G17) and the position of the contour in Z direction. The spindle angle (C) defines the spindle position.

Parameters

X Area diameter (as cutting limit)

Z Position of the reference plane—default: 0

C Spindle angle—default: 0

LATERAL_Y section

The section code identifies the YZ plane (G19). For machines equipped with a B axis, it defines the tilted plane.

Without tilted plane: The reference diameter defines the contour position in the X direction; the C axis angle defines the position on the workpiece.

Parameters

X	Reference diameter
C	C axis angle—Defines the spindle position

With tilted plane (see figures): SURFACE_Y additionally performs the following transformations and rotations for the tilted plane:

- Shifts the coordinate system to the position I, K
- Rotates the coordinate system by the angle B; reference point: I, K
- H=0: Shifts the rotated coordinate system by –I. The coordinate system is moved "back."

Parameters

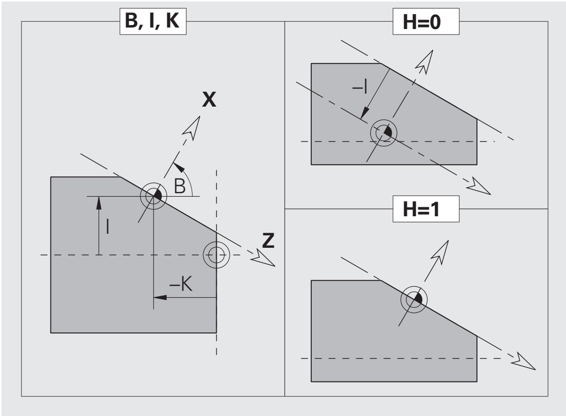
X	Reference diameter
C	C axis angle—Defines the spindle position
B	Plane angle: Positive Z axis
I	Plane reference in X direction (radius)
K	Plane reference in Z direction
H	Automatic shift of the coordinate system (default: 0) ■ 0: The rotated coordinate system is shifted by –I ■ 1: The coordinate system is not shifted

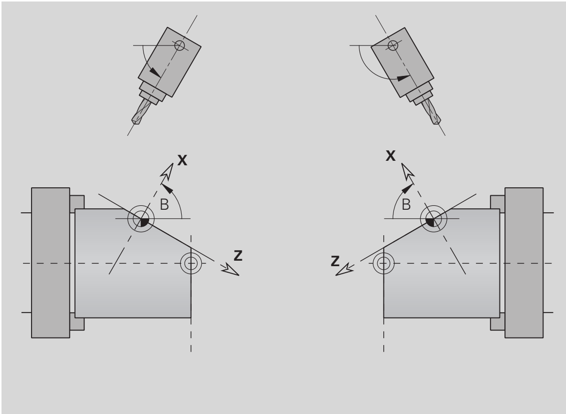
Shifting "back" coordinate system: The control evaluates the reference diameter for the cutting limit. This value is also used as the reference value for the depth that you program for drilling operations and milling contours.

Since the reference diameter is referenced to the current zero point, it is recommended when working in a tilted plane, to shift the rotated coordinate system "back" by the distance –I. If the cutting limits are not needed, for example for drilling holes, you can disable the shift of the coordinate system (H=1) and set the reference diameter to 0.

Please note:

- X is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
- Mirroring the coordinate system has no effect on the reference axis of the tilt angle ("B axis angle" of the tool call).





Example: "SURFACE_Y"

```
HEADER
...
CONTOUR Q1 X0 Z600
BLANK
...
FINISHED PART
...
SURFACE_Y X118 C0 B130 I59 K0
...
MACHINING
...
```

MACHINING section

In the **MACHINING** program section you program the machining operations. This code **must** be included.

END code

The **END** code concludes the NC program. This code **must** be included.

SUBPROGRAM section

If you define a subprogram within your NC program (within the same file), it is designated with **SUBPROGRAM**, followed by the name of the subprogram (max. 40 characters).

RETURN code

The RETURN code concludes the subprogram.

CONST code

In the CONST section of the program you define constants. You use constants for the definition of a value.

You enter the value directly or you calculate it. If you use constants in the calculation you must first define them.

The length of the constant name must not exceed 20 characters. Lower case letters and numbers are allowed. Constants always begin with an underscore: See "Expanded variable syntax CONST – VAR" on page 398.

Example: CONST

```
CONST
_nvr = 0
_sd=PARA("","CfgGlobalTechPara","safetyDis
tWorkpOut")
_nws = _sd-_nvr
. . .
BLANK
N 1 G20 X120 Z_nws K2
. . .
MACHINING
N 6 G0 X100+_sd
. . .
```

VAR code

In the **VAR** program section, you assign names (descriptive text) to variables: See "Expanded variable syntax CONST – VAR" on page 398..

The length of the variable name must not exceed 20 characters. Lower case letters and numbers are allowed. Variables always begin with "#".

Example: VAR

```
VAR
#_inside_dm = #l2
#_length = #g3
. . .
BLANK
N 1 #_length=120
N 2 #_inside_dm=25
N 3 G20 X120 Z#_length+2 K2 I#_inside_dm
. . .
MACHINING
. . .
```

1.4 Tool programming

The designations of the tool pockets are fixed by the machine tool builder. Each tool holder has a unique **T number.**

In the T command (MACHINING section) you program the T number, and therefore the position to which the tool carrier rotates. The Control retrieves the assignment of the tools to the turret position from the turret list of the TURRET section.

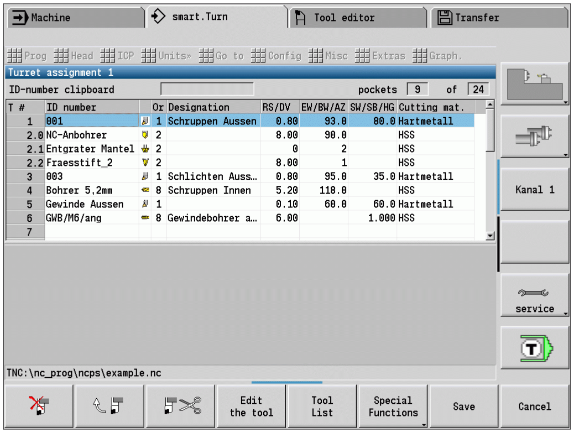
You can edit the tool entries individually, or you can call the tool list via the `Set up the turret list` menu item and then edit it.

Setting up a tool list

In the "Set up the turret list" function, the Control provides the turret assignment as a tool list for editing.

You have the following options:

- Editing the turret assignment: Transfer tools from the database, delete entries or move them to other positions (for soft keys see table).
- Loading the turret list of the Machine mode of operation.
- Deleting the current turret assignment of the NC program.



Loading the turret list of the Machine mode of operation:

- Select "Head > Set up the turret list".

`Special Functions`

- Switch to "Special functions":

`Transfer machine`

- Load the tool list of the Machine mode of operation into the NC program.

Deleting a tool list:

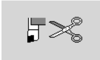
- Select "Head > Set up the turret list".

`Special Functions`

- Switch to "Special functions":

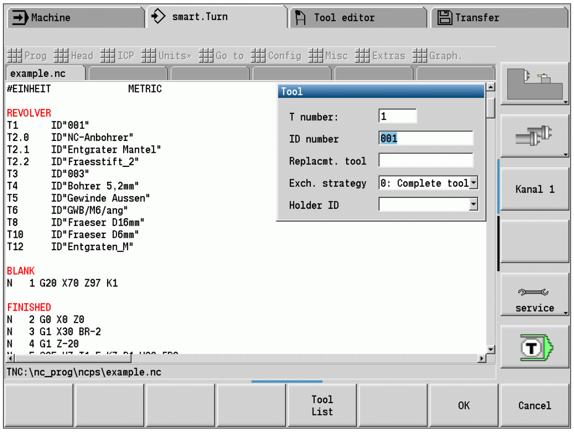
`Delete all`

- Delete all entries of the turret list.

Soft keys in turret list	
	Delete entry
	Paste entry from clipboard
	Cut out entry and save it in the clipboard
Tool list	Show entries in the tool database
Save	Save the turret assignment
Cancel	Close the tool list. You decide whether the changes made remain in effect
Editing	The input window of the selected tool is opened for editing

Editing tool entries

For each entry of the TURRET section you call the Tool dialog box, enter the identification number or use the identification number from the tool database.



New tool entry

INS

Position the cursor and press the INS (insert) key. The editor opens the Tool dialog box.

Enter the identification number of the tool.

Tool list

Open the tool database.

Place the cursor on the tool to be loaded.

Load tool

Transfer the identification number of the tool.

Editing the tool data

Position the cursor on the entry to be edited and press RETURN.

Edit the Tool dialog box.

Parameters of the "Tool" dialog box

T number	Position on tool carrier
ID number	ID number (reference to database)
Replacement tool	Identification number of the tool to be used when the previous tool is worn out.
Replacement strategy	■ 0: Complete tool ■ 1: Secondary cutting edge or any

Multipoint tools

A multipoint tool is a tool with multiple reference points or multiple cutting edges. During T call, the T number is followed by an S to identify the cutting edge.

T number.S (S=0 to 9)

S=0 identifies the main cutting edge, which does not need to be programmed.

Examples:

- T3 or T3.0: Tilted position 3; main cutting edge
- T12.2: Tilted position 12; cutting edge 2

Replacement tools

During "simple" tool life monitoring the MANUALplus stops program run when a tool is worn out. However, the program run is then resumed and concluded.

If you use **the tool life monitoring with replacement tools** function, the Control automatically inserts the "sister tool" as soon as the tool is worn out. The Control does not stop the program run until the last tool of the tool sequence of exchange is worn out.

You can define replacement tools when setting up the turret. The "interchange chain" can contain more than one replacement tool. The interchange chain is a part of the NC program.

In the T commands, you program the first tool to be changed.

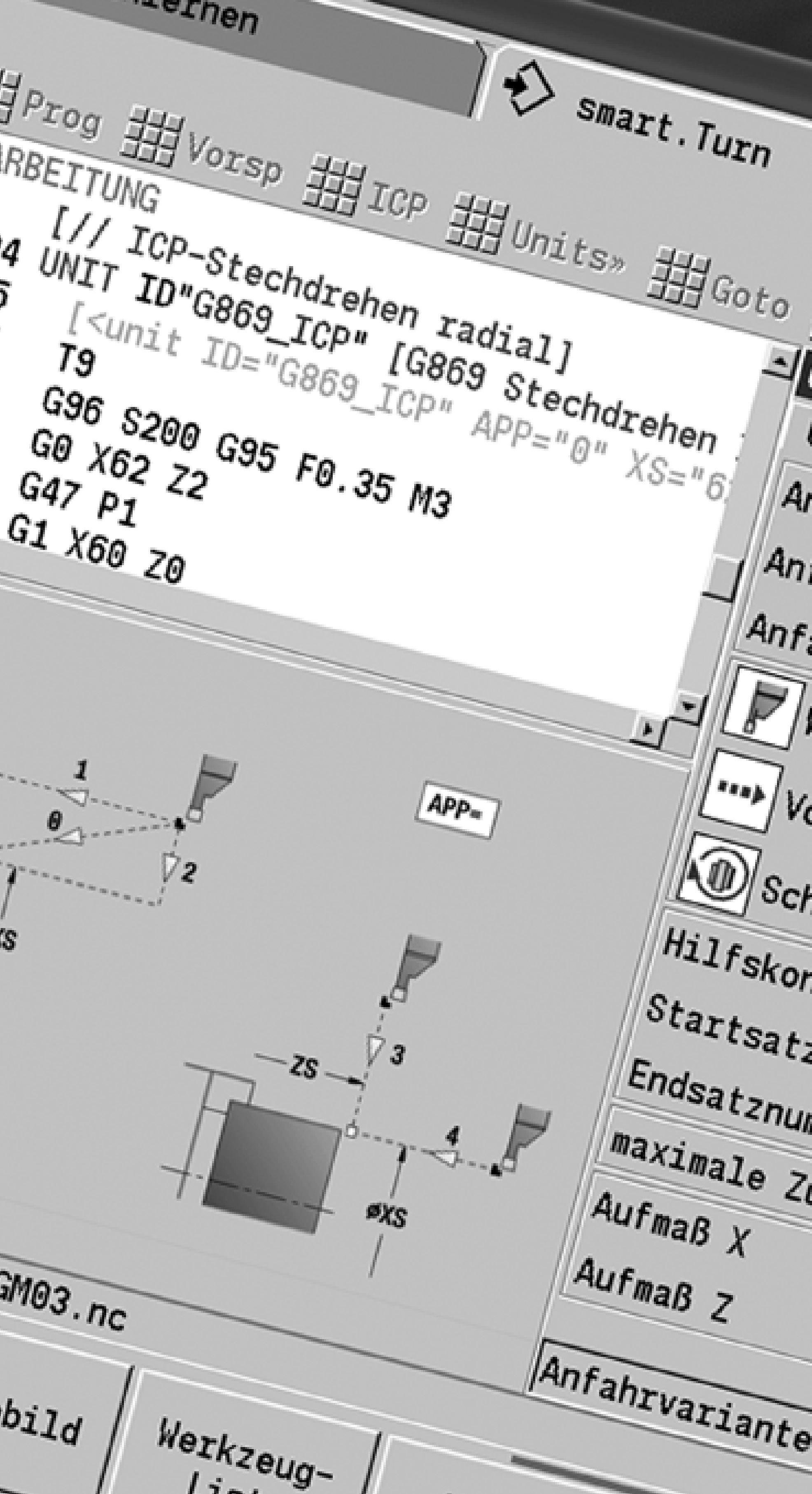
Defining replacement tools:

Place the cursor on the previous tool and press RETURN.

Enter the identification number of the replacement tool (Tool dialog box) and define the replacement strategy.

When using multipoint tools, you define in the **replacement strategy** whether the complete multipoint tool or only the worn-out cutting edge of the tool is to be replaced by a replacement tool:

- 0: Complete tool (default): If a cutting edge of the multipoint tool is worn out, the tool will no longer be used.
- 1: Secondary cutting edge or any: Only the worn-out cutting edge of the multipoint tool is replaced by another tool or another cutting edge. Any other cutting edges of the multipoint tool that are not worn out will continue to be used.



2

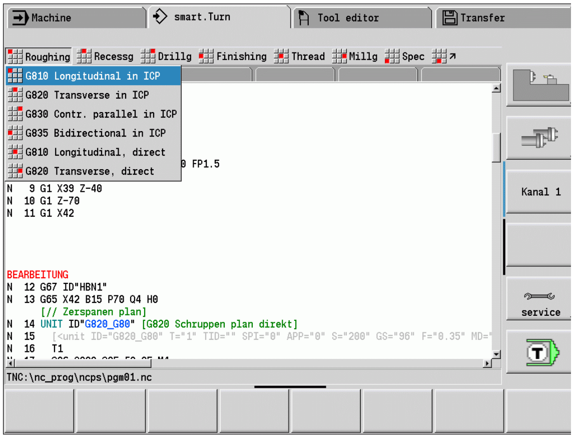
smart.Turn units

2.1 smart.Turn units

"Units" menu

The **"Units" menu** contains the unit calls grouped by the type of machining operation: Select the Units menu to call the following pull-down menus:

- Roughing
- Recessing
- Drilling and predrilling (C axis and Y axis)
- Finishing
- Thread
- Milling (C axis and Y axis)
- Special operations

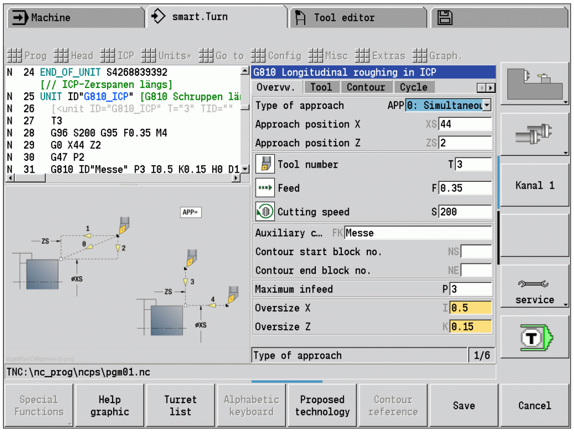


The smart.Turn unit

A unit describes a complete working block. This means that the unit includes the tool call, the technology data, the cycle call, the approach and departure strategies as well as global data, such as safety clearance, etc. All of these parameters are collected in one, clearly structured dialog box.

Unit forms

The unit dialog is divided into fillable forms and the forms are divided again into groups. You can navigate between the forms and groups with the **smart keys**.



Forms in unit dialogs

	Overvw. / Tool / Contour / Cycle
`Overview`	Overview form with all necessary settings.
`Tool`	Tool form with tool selection, technological settings and M functions.
`Contour`	Description or selection of the contour to be machined
`Cycle`	Description of the machining operation
`Global`	View and settings of globally set values
`AppDep`	Definition of approach and departure behavior
`ToolExt`	Extended tool settings

The Overview form

The overview form summarizes the most important settings of the unit. These parameters are repeated in the other forms.

The Tool form

You program the technological information in this form.

Tool form

Tool

T	Tool number (number of turret pocket).
TID	The identification number (tool name) is entered automatically.
F	Feed rate: Feed rate in revolutions for machining (mm/rev). The tool is moved at the programmed value for each spindle revolution.
S	(Constant) cutting speed (m/min) or constant shaft speed (rev/min). Switchable with **`Type of turning GS`**.

Spindle

GS Type of turning

- G96: Constant surface speed The rotational speed changes with the turning diameter.
- G97: Constant shaft speed. Rotational speed is independent of the turning diameter.

MD Direction of rotation

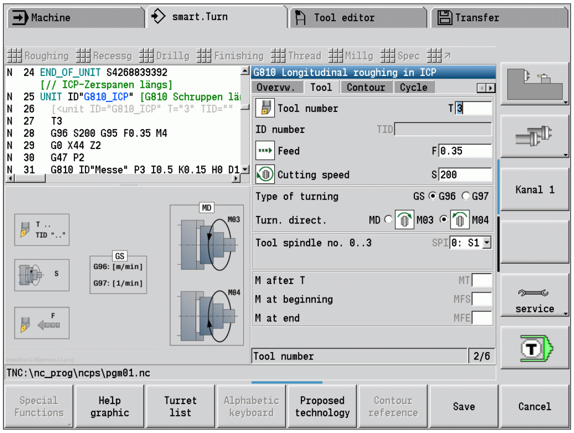
- M03: Clockwise (CW)
- M04: Counterclockwise (CCW)

SPI	Workpiece spindle number (0 to 3). Spindle that is holding the workpiece (only on machines with more than one spindle).
SPT	Tool spindle number (0 to 3). Spindle of the driven tool.

M functions

MT	M after T: M function that is executed after the tool call T.
MFS	M at beginning: M function that is executed at the beginning of the machining step.
MFE	M at end: M function that is executed at the end of the machining step.

A machining operation is assigned to each unit for access to the technology database. The following description shows the assigned machining mode and the unit parameters that were changed by the technology proposal.



Soft keys in the tool form

Tool list	Selects the tool number
Proposed technology	Loads the feed rate, cutting speed and infeed from the **technology database**.

The Contour form

In the contour form you define the contours to be machined. A difference is made between the direct contour definition (G80) and the reference to an **external** contour definition (FINISHED part or AUXIL_CONTOUR sections).

ICP contour definition parameters

FK Auxiliary contour: Name of the contour to be machined.

You can select an existing contour or describe a new contour with ICP.

NS Contour start block number: Beginning of contour section.

NE Contour end block number: End of contour section.

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

V Machine form elements (default: 0).

A chamfer/rounding arc is machined:

- 0: At start and end of the contour
- 1: At start of the contour
- 2: At end of the contour
- 3: No machining
- 4: Only chamfer/rounding is machined—not the base element. (requirement: the contour section consists of a single element)

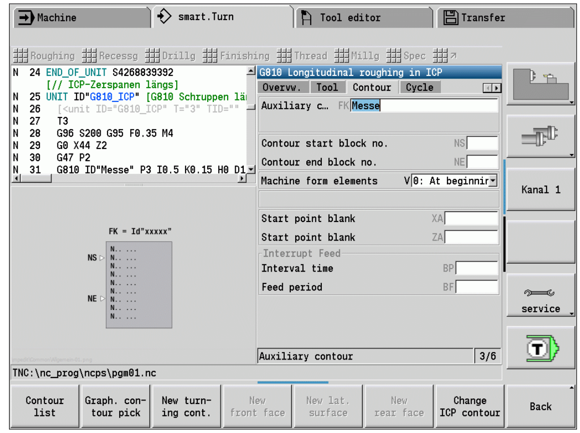
XA, ZA Starting point of blank (only effective if no blank was programmed):

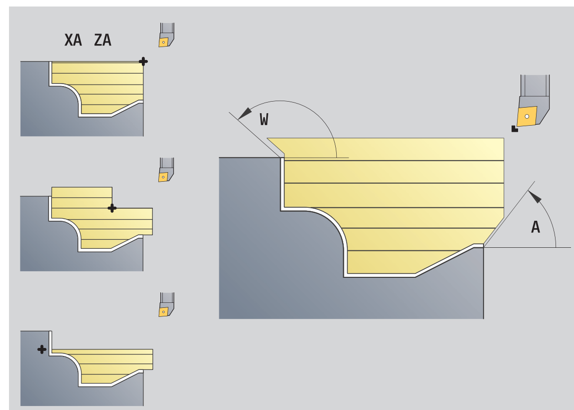
- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the workpiece blank.

BP Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.

BF Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.

The listed soft keys are **only** selectable if the input cursor is in the FK field, or on **NS** or **NE**.





Soft keys in the ICP contour form

Contour list	Opens the selection list of the contours defined in the program.
Graph. con- tour pick	Shows all contours in the graphics window. Use the arrow keys for selection.
New turn- ing cont.	Starts the ICP editor. First, enter the desired contour name in **FK**.
Change ICP contour	Starts the ICP editor with the currently selected contour.
Contour reference	Opens the graphics window for selection of a part of a contour for **NS** and **NE**.

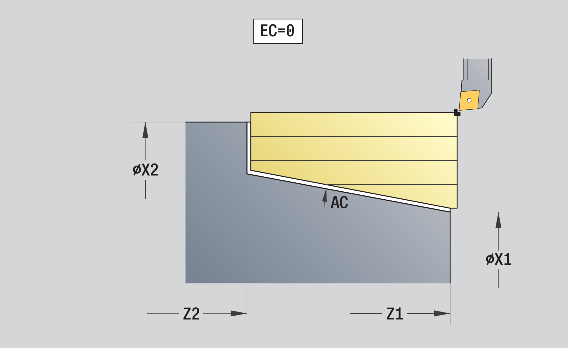
Direct contour definition parameters for turning operations

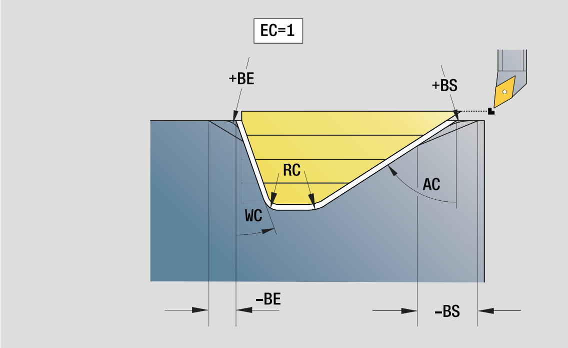
EC Type of contour
- 0: Normal contour
- 1: Plunging contour

X1, Z1 Contour starting point
X2, Z2 Contour end point
RC Rounding: Radius of contour corner
AC Start angle: Angle of the first contour element (range: 0° < 90°)
WC End angle: Angle of the last contour element (range: 0° < 90°)
BS –Chamfer/+radius at start:
- BS>0: Radius of rounding arc
- BS<0: Section length of chamfer

BE –Chamfer/+radius at end:
- BE>0: Radius of rounding arc
- BS<0: Section length of chamfer

BP Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.

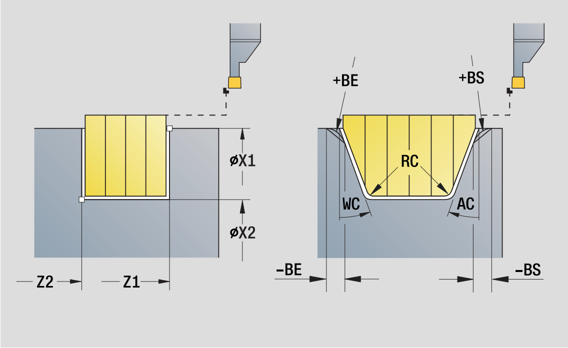




Direct contour definition parameters for recessing operations

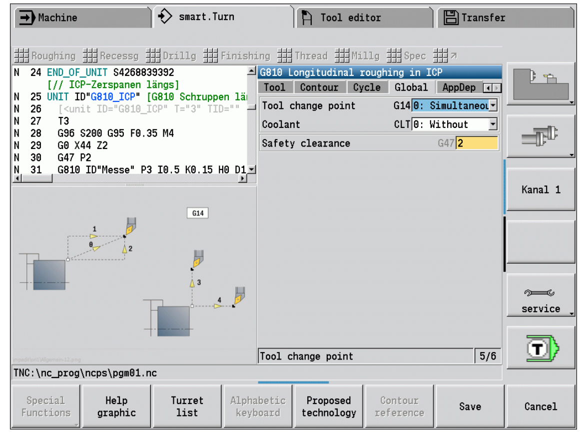
X1, Z1 Contour starting point
X2, Z2 Contour end point
RC Rounding: Radii in the recess base
AC Start angle: Angle of the first contour element (range: 0° < 90°)
WC End angle: Angle of the last contour element (range: 0° < 90°)
BS –Chamfer/+radius at start:
- BS>0: Radius of rounding arc
- BS<0: Section length of chamfer

BE –Chamfer/+radius at end:
- BE>0: Radius of rounding arc
- BS<0: Section length of chamfer



The Global form

This form contains parameters that were defined as default values in the start unit. You can edit these parameters in the machining units.



Parameters on the Global form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

G47 Safety clearance: Indicates the distance to the current blank material at which the tool is **not** moved at rapid traverse.

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

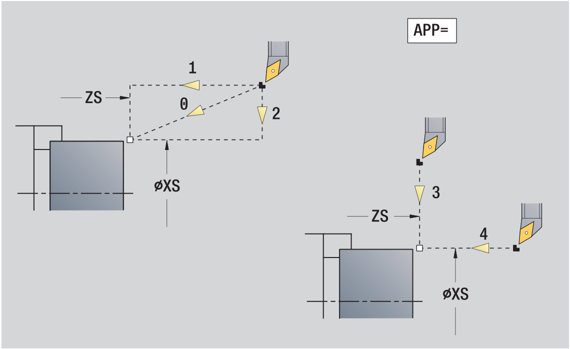
SCI Safety clearance in plane: Safety clearance in the working plane during drilling and milling operations.

G60 Protection zone. During drilling and boring the protection zone monitoring is

- 0: Active
- 1: Inactive

The AppDep form

Positions and variants of the approach and departure movements are defined in this form.



Approach: Influence the approach strategy.

"Approach" parameters

APP Type of approach:

- No axis (switch off the approach function)
- 0: Simultaneous (X and Z axes approach diagonally)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z

XS, ZS Approach position: Position of the tool point before cycle call

Additionally with C-axis operations:

CS Approach position: C-axis position that is approached before cycle call with G110.

"Approach with Y axis" parameters

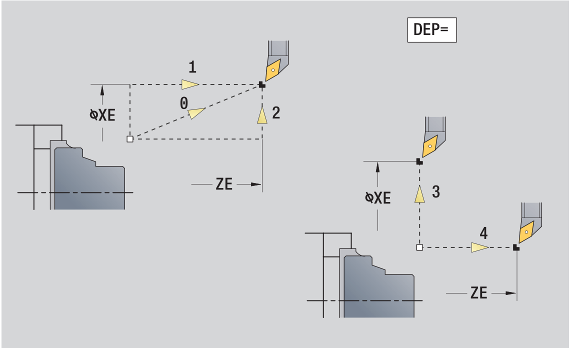
APP Type of approach:

- No axis (switch off the approach function)
- 0: Simultaneous (X and Z axes approach diagonally)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes approach diagonally)

XS, YS, ZS Approach position: Position of the tool point before cycle call

CS Approach position: C-axis position that is approached before cycle call with G110.

Departure: Influence the departure strategy (also applies for Y-axis functions).



"Departure" parameters

DEP Type of departure:

- No axis (switch off the departure function)
- 0: Simultaneous (X and Z axes depart diagonally)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z

XE, ZE Departure position: Position of the tool point before the movement to the tool change point.

The Tool Ext form

In this form you can program additional tool settings.

Tool Ext form
Tool

T	Tool number (number of turret pocket).
TID	The identification number (tool name) is entered automatically.

B axis

B	Angle in the B axis (machine-dependent function)
CW	C tilting plane angle: Position of the C axis to determine the work position of the tool (machine-dependent function)

Miscellaneous functions

HC Shoe brake (machine-dependent function)

- 0: Automatic
- 1: Tighten
- 2: Don't tighten

DF	Additional function: Can be evaluated by the machine manufacturer in a subprogram (machine-dependent function)
XL, ZL, YL	Values can be evaluated by the machine manufacturer in a subprogram (machine-dependent function)

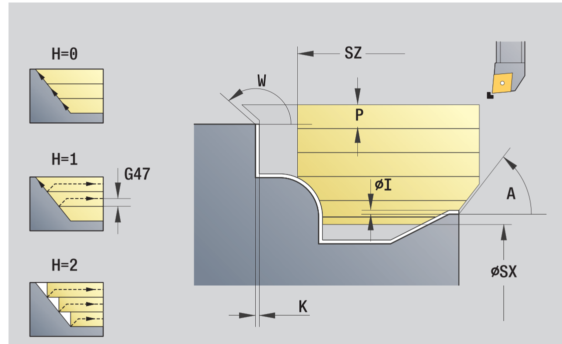
With the **`Advanced T change`** soft key you can switch quickly and easily between the **`Tool`** and **`Tool Ext`** forms.

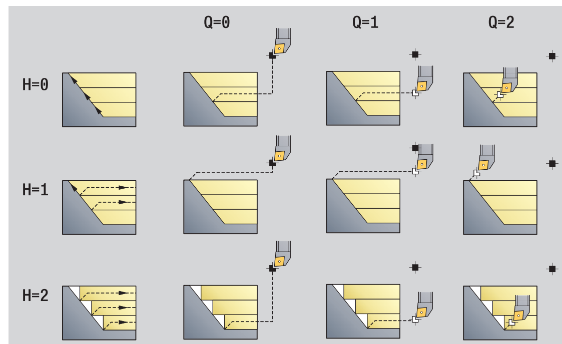
2.2 Units—Roughing

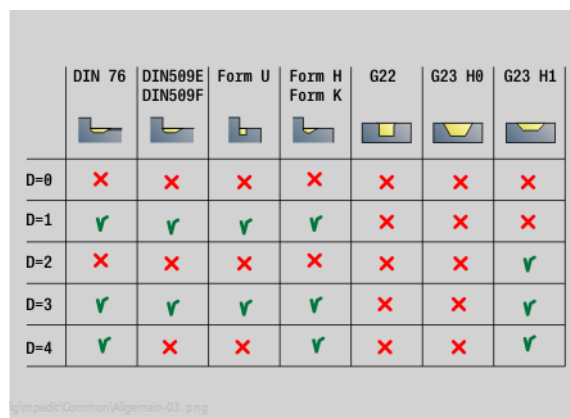
"Longitudinal roughing in ICP" unit

The unit machines the contour described in the FINISHED program section from "NS to NE". Any auxiliary contour defined in FK will be used.

Unit name: G810_ICP / Cycle: G810 (see page 264)







	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

Contour form: see page 60

Cycle form

I, K — Oversize in X, Z direction (I: diameter value)

P — Maximum infeed

E — Plunging behavior

- E=0: Descending contours are not machined
- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

SX, SZ — Cutting limit (SX: diameter value)—(default: no cutting limit)

A — Approach angle (reference: Z-axis)—(default: parallel to Z axis)

W — Departure angle (reference: Z axis)—(default: orthogonal to Z axis)

Q — Type of retraction at end of cycle

- 0: Returns to starting point, first X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

H — Contour smoothing

- 0: With each cut along the contour (within the infeed range)
- 1: Contour smoothing with the last cut (entire contour); retracts at 45%
- 2: No smoothing; retracts at 45°

D — Omit elements (see figure)

O — Hide undercutting:

- 0: Undercuts are machined
- 1: Undercuts are not machined

Further forms: see page 58

Access to the technology database:

- Machining operation: Roughing
- Affected parameters: F, S, E, P

"Transverse roughing in ICP" unit

The unit machines the contour described in the FINISHED program section from "NS to NE". Any auxiliary contour defined in FK will be used.

Unit name: G820_ICP / Cycle: G820 (see page 267)

Contour form: see page 60

Cycle form

I, K — Allowance in X and Z direction (I = diameter)

P — Maximum infeed

E — Plunging behavior

- E=0: Descending contours are not machined
- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

SX, SZ — Cutting limit (SX: diameter value)—(default: no cutting limit)

A — Departure angle (reference: Z axis)—(default: orthogonal to Z axis)

W — Departure angle (reference: Z-axis)—(default: parallel to Z axis)

Q — Type of retraction at end of cycle

- 0: Returns to starting point, first X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

H — Contour smoothing

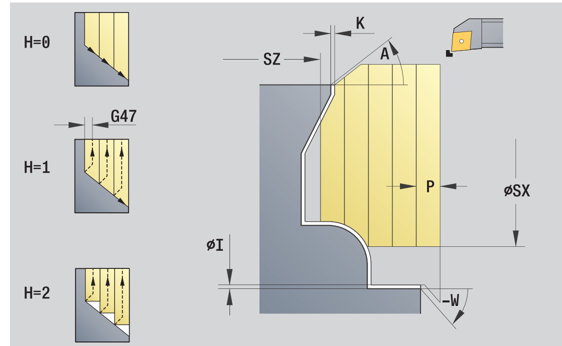
- 0: With each cut along the contour (within the infeed range)
- 1: Contour smoothing with the last cut (entire contour); retracts at 45%
- 2: No smoothing; retracts at 45°

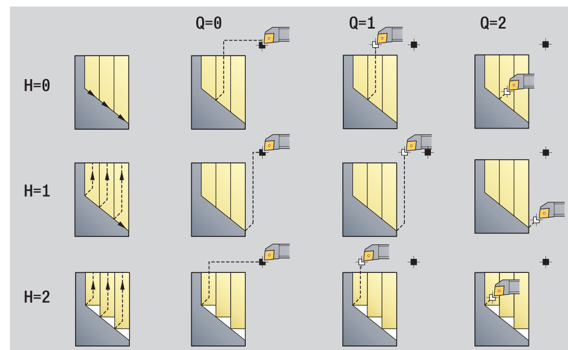
D — Hide elements; do not machine form elements (see figure)

O — Hide undercutting:

- 0: Undercuts are machined
- 1: Undercuts are not machined

Further forms: see page 58





	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

Access to the technology database:

- Machining operation: Roughing
- Affected parameters: F, S, E, P

"Contour-parallel roughing in ICP" unit

The unit machines the contour described in the FINISHED program section from "NS to NE" parallel to the contour. Any auxiliary contour defined in FK will be used.

Unit name: G830_ICP / Cycle: G830 (see page 270)

Contour form

J — Workpiece blank oversize (radius value)—active only if **no blank** has been defined.

B — Contour calculation

- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

Further parameters of the contour form: see page 60.

Cycle form

P — Maximum infeed

I, K — Oversize in X, Z direction (I: diameter value)

SX, SZ — Cutting limit (SX: diameter value)—(default: no cutting limit)

A — Approach angle (reference: Z axis)—(default: parallel to Z axis)

W — Departure angle (reference: Z axis)—(default: orthogonal to Z axis)

Q — Type of retraction at end of cycle

- 0: Returns to starting point, first X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

H — Type of cut lines (cutting paths)

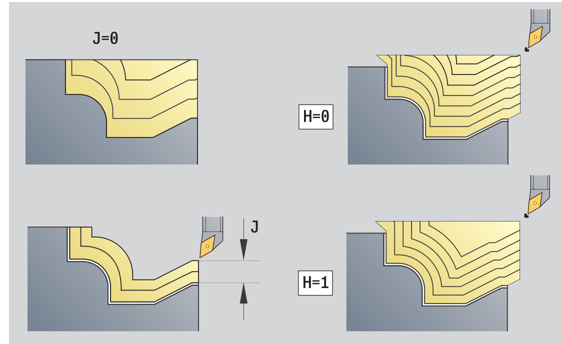
- 0: Constant cutting depth: Contour is shifted by a constant infeed value (paraxial)
- 1: Equidistant cutting lines: Cutting lines run at a constant distant from the contour (contour parallel). The contour is scaled.

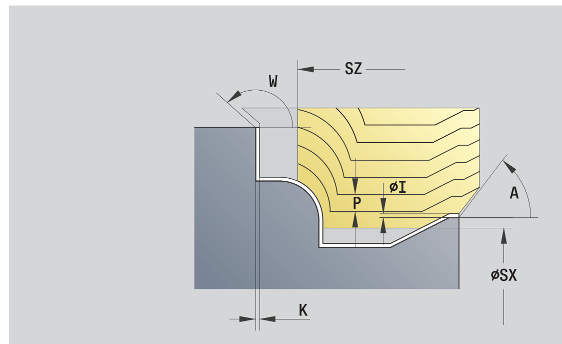
HR — Main machining direction

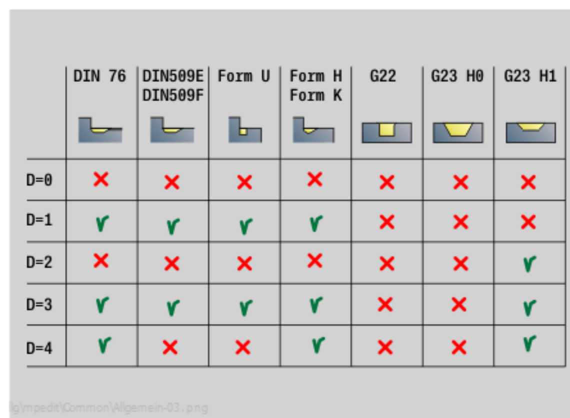
- 0: Automatic
- 1: +Z
- 2: +X
- 3: –Z
- 4: –X

D — Hide elements; do not machine form elements (see figure)

Further forms: see page 58







	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

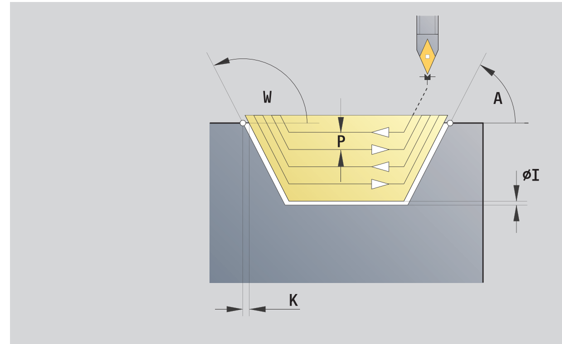
Access to the technology database:

- Machining operation: Roughing
- Affected parameters: F, S, E, P

"Bidirectional roughing in ICP" unit

The unit machines the contour described in the FINISHED program section from "NS to NE" parallel to the contour and bidirectionally. Any auxiliary contour defined in FK will be used.

Unit name: G835_ICP / Cycle: G835 (see page 273)



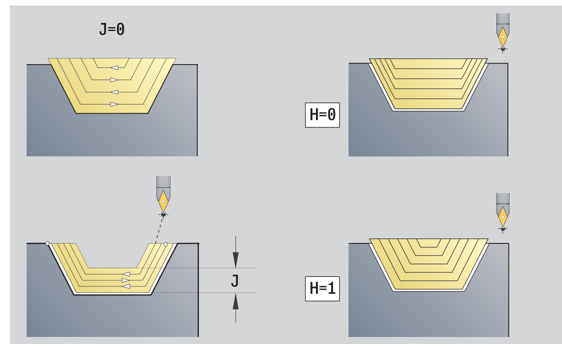
Contour form

J Workpiece blank oversize (radius value)—active only if **no blank** has been defined.

B Contour calculation

- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

Further parameters of the contour form: see page 60.



Cycle form

P Maximum infeed

I, K Allowance in X and Z direction (I = diameter)

SX, SZ Cutting limit (SX: diameter value)—(default: no cutting limit)

A Approach angle (reference: Z axis)—(default: parallel to Z axis)

W Departure angle (reference: Z axis)—(default: orthogonal to Z axis)

Q Type of retraction at end of cycle

- 0: Returns to starting point, first X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

H Type of cut lines (cutting paths)

- 0: Constant cutting depth: Contour is shifted by a constant infeed value (paraxial)
- 1: Equidistant cutting lines: Cutting lines run at a constant distant from the contour (contour parallel). The contour is scaled.

D Hide elements; do not machine form elements (see figure)

Further forms: see page 58

	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

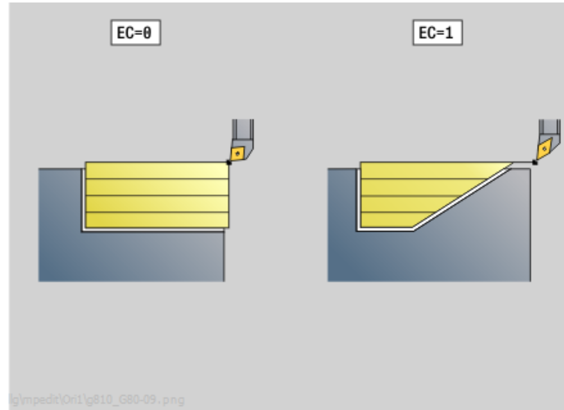
Access to the technology database:

- Machining operation: Roughing
- Affected parameters: F, S, E, P

"Longitudinal roughing with direct contour input" unit

The unit machines the contour defined by the parameters. In **EC** you define whether you want to machine a normal or a plunging contour.

Unit name: G810_G80 / Cycle: G810 (see page 264)



Contour form

EC Type of contour

- 0: Normal contour
- 1: Plunging contour

X1, Z1 Contour starting point

X2, Z2 Contour end point

RC Rounding: Radius of contour corner

AC Start angle: Angle of the first contour element (range: 0° < 90°)

WC End angle: Angle of the last contour element (range: 0° < 90°)

BS –Chamfer/+radius at start:

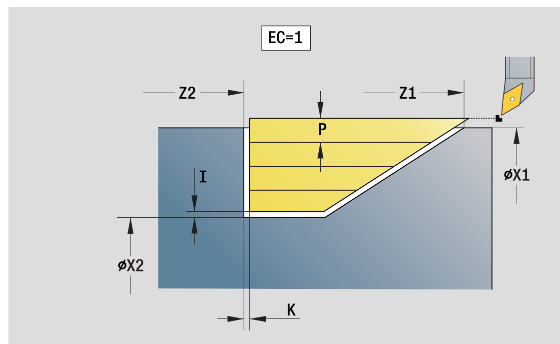
- BS>0: Radius of rounding arc
- BS<0: Section length of chamfer

BE –Chamfer/+radius at end

- BE>0: Radius of rounding arc
- BS<0: Section length of chamfer

BP Break duration: Time span for interruption of the feed for chip breaking.

BF Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.



Cycle form

P Maximum infeed

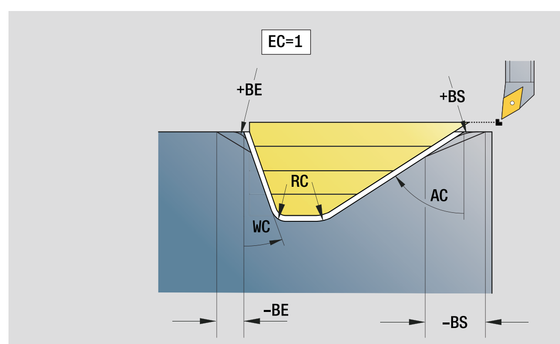
I, K Oversize in X, Z direction (I: diameter value)

E Plunging behavior

- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

H Contour smoothing

- 0: With each cut along the contour (within the infeed range)
- 1: Contour smoothing with the last cut (entire contour); retracts at 45%
- 2: No smoothing; retracts at 45°



Further forms: see page 58

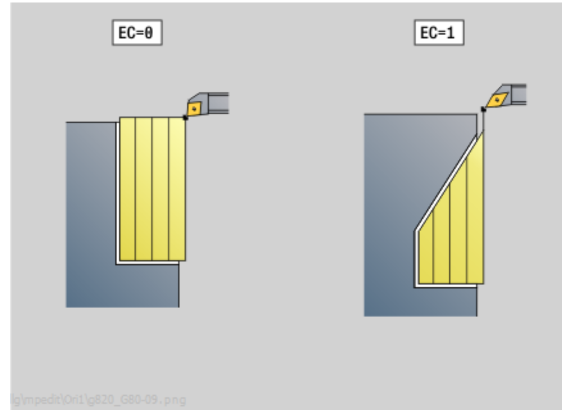
Access to the technology database:

- Machining operation: Roughing
- Affected parameters: F, S, E, P

"Transverse roughing with direct contour input" unit

The unit machines the contour defined by the parameters. In **EC** you define whether you want to machine a normal or a plunging contour.

Unit name: G820_G80 / Cycle: G820 (see page 267)



Contour form

EC Type of contour

- 0: Normal contour
- 1: Plunging contour

X1, Z1 Contour starting point

X2, Z2 Contour end point

RC Rounding: Radius of contour corner

AC Start angle: Angle of the first contour element (range: 0° < AC < 90°)

WC End angle: Angle of the last contour element (range: 0° < WC < 90°)

BS Chamfer/radius at start

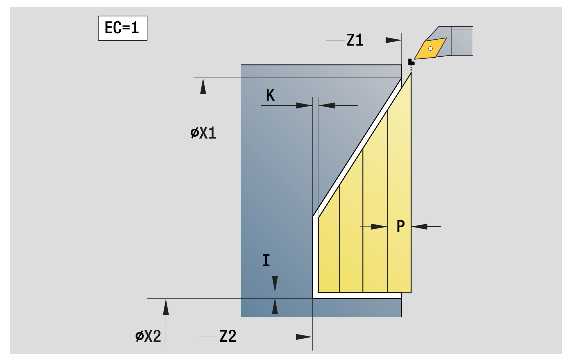
- BS>0: Radius of rounding arc
- BS<0: Section length of chamfer

BE Chamfer/radius at end

- BE>0: Radius of rounding arc
- BS<0: Section length of chamfer

BP Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.

BF Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.



Cycle form

P Maximum infeed

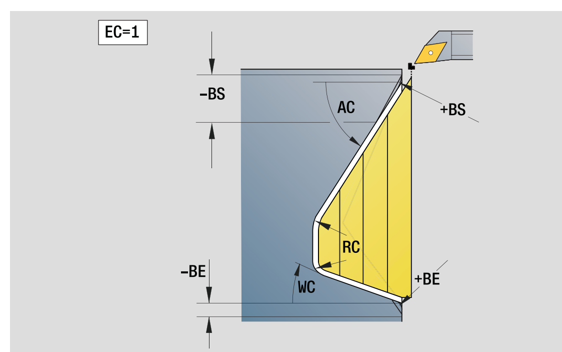
I, K Oversize in X, Z direction (I: diameter value)

E Plunging behavior

- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

H Contour smoothing

- 0: With each cut along the contour (within the infeed range)
- 1: Contour smoothing with the last cut (entire contour); retracts at 45%
- 2: No smoothing; retracts at 45°



Further forms: see page 58

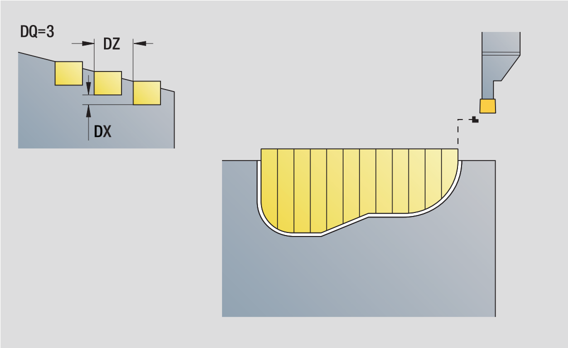
Access to the technology database:

- Machining operation: Roughing
- Affected parameters: F, S, E, P

2.3 Units—Recessing

"ICP contour recessing" unit

The unit machines the contour described in the FINISHED program section axially/radially from "NS to NE". Any auxiliary contour defined in FK will be used.



Unit name: G860_ICP / Cycle: G860 (see page 275)

Contour form

DQ Number of recessing cycles

DX, DZ Distance to subsequent recess in X, Z direction (DX: radius value)

Further parameters of the contour form: see page 60.

Cycle form

I, K Oversize in X, Z direction (I: diameter value)

SX, SZ Cutting limit (SX: diameter value)—(default: no cutting limit)

ET Recessing depth by which one cut is fed.

P Cutting width (recessing width): (default: 0.8 x tool width)

E Finishing feed rate. Differing feed rate used only for the finishing process.

EZ Period of dwell after recessing path (default: time for one spindle revolution)

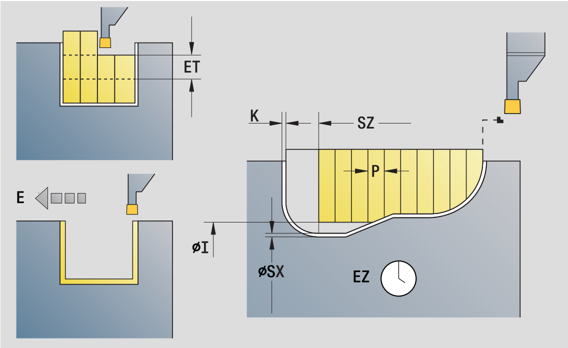
Q Roughing/finishing (process variants)

- 0 (SS): Roughing and finishing
- 1 (SP): Only roughing
- 2 (SL): Only finishing

H Type of retraction at end of cycle

- 0: Return to starting point
  - Axial recess: First Z, then X direction
  - Radial recess: First X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

Further forms: see page 58



Access to the technology database:

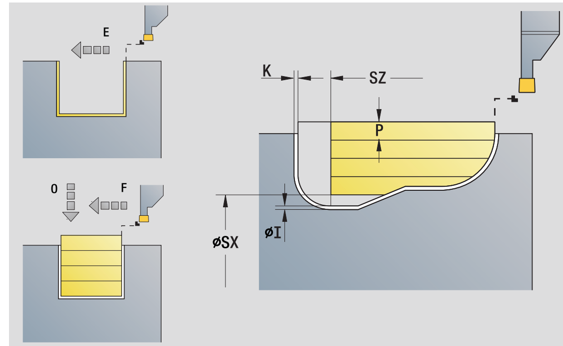
- Machining operation: Contour recessing
- Affected parameters: F, S, E

"ICP recess turning" unit

The unit machines the contour described by ICP axially/radially from "NS to NE". The workpiece is machined by alternate recessing and roughing movements.

The unit machines the contour described in the FINISHED program section axially/radially from "NS to NE". Any auxiliary contour defined in FK will be used.

Unit name: G869_ICP / Cycle: G869 (see page 278)



Contour form

X1, Z1 Starting point of blank. Evaluation only if no blank has been defined.

RI, RK Workpiece blank oversize in X and Z direction

SX, SZ Cutting limit (SX: diameter value)—(default: no cutting limit)

Further parameters of the contour form: see page 60.

Cycle form

P Maximum infeed during rough turning

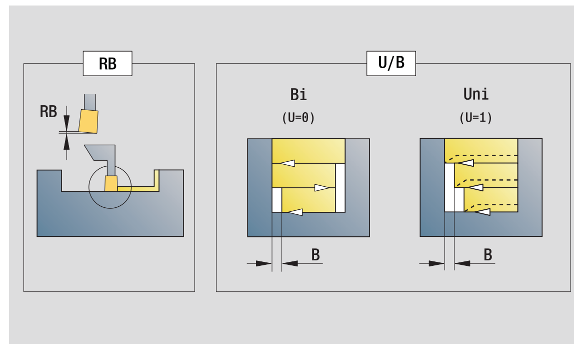
I, K Oversize in X, Z direction (I: diameter value)

RB Turning depth compensation for finishing

B Offset width

U Cutting direction

- 0 (Bi): Bidirectional (in both directions)
- 1 (Uni): Unidirectional (in direction of contour)



Q Sequence (roughing/finishing)

- 0: Roughing and finishing
- 1: Only roughing
- 2: Only finishing

A Approach angle (default: opposite to recessing direction)

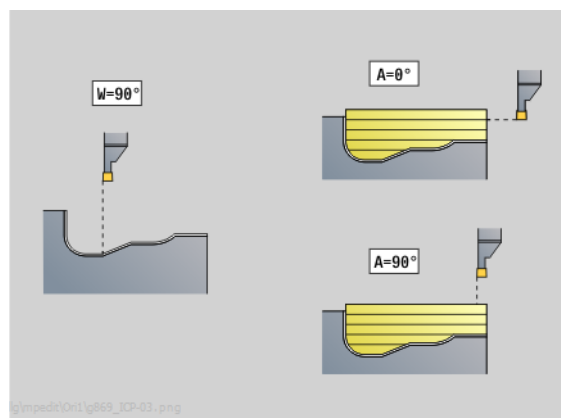
W Departure angle (default: opposite to recessing direction)

O Recessing feed rate (default: active feed rate)

E Finishing feed rate (default: active feed rate)

H Type of retraction at end of cycle

- 0: Return to starting point
  - Axial recess: First Z, then X direction
  - Radial recess: First X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops



Further forms: see page 58

The Control uses the tool definition to distinguish between radial and axial recessing.

Turning depth compensation RB: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor. The value is usually determined empirically.

Access to the technology database:

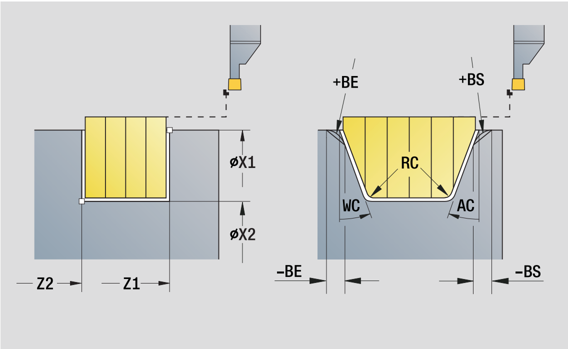
- Machining operation: Recess turning
- Affected parameters: F, S, O, P

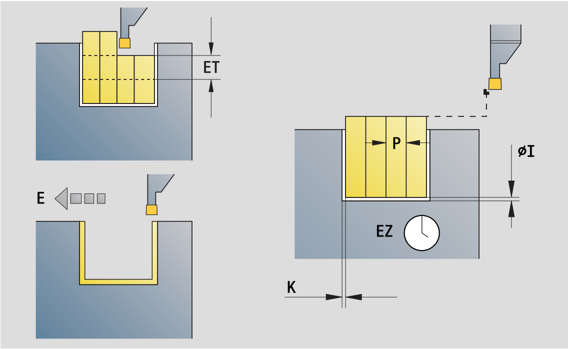
Offset width B: After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the offset width B. Each time the system switches on this side, the path is reduced by B—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius). If required, the Control reduces the programmed offset width. After clearance roughing, the remaining material is removed with a single cut.

"Contour recessing with direct contour input" unit

The unit machines the contour defined by the parameters axially/ radially.

Unit name: G860_G80 / Cycle: G860 (see page 275)





Contour form:

RI, RK Workpiece blank oversize in X and Z direction

Further parameters of the contour form: see page 60.

Cycle form

Q Roughing/finishing (process variants)

- 0: Roughing and finishing
- 1: Only roughing
- 2: Only finishing

I, K Oversize in X, Z direction (I: diameter value)

P Cutting width (recessing width): (default: 0.8 x tool width)

E Finishing feed rate: Differing feed rate used only for the finishing process

EZ Period of dwell after recessing path (default: time for one spindle revolution)

DQ Number of recessing cycles

DX, DZ Distance to subsequent recess in X, Z direction

Further forms: see page 58

The Control uses the tool definition to distinguish between radial and axial recessing.

Access to the technology database:

- Machining operation: Contour recessing
- Affected parameters: F, S, E

"Recess turning with direct contour input" unit

The unit machines the contour defined by the parameters axially/radially. The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

Unit name: G869_G80 / Cycle: G869 (see page 278)

Contour form:

RI, RK   Workpiece blank oversize in X and Z direction

Further parameters of the contour form: see page 60.

Cycle form

P   Maximum infeed during rough turning

I, K   Oversize in X, Z direction (I: diameter value)

RB   Turning depth compensation for finishing

B   Offset width

U   Cutting direction

- 0 (Bi): Bidirectional (in both directions)
- 1 (Uni): Unidirectional (in direction of contour)

Q   Sequence (roughing/finishing)

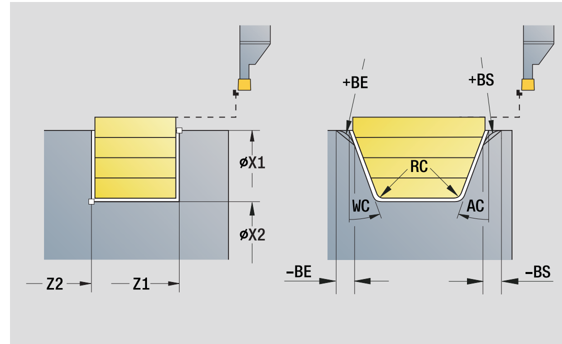
- 0: Roughing and finishing
- 1: Only roughing
- 2: Only finishing

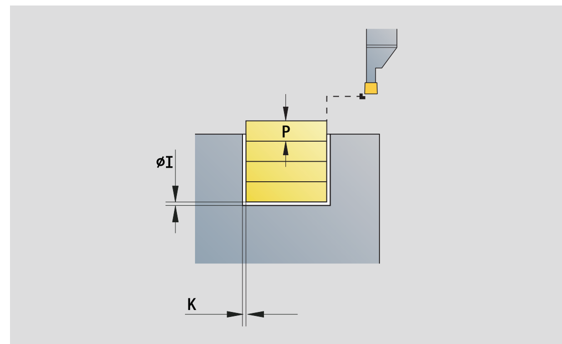
Further forms: see page 58

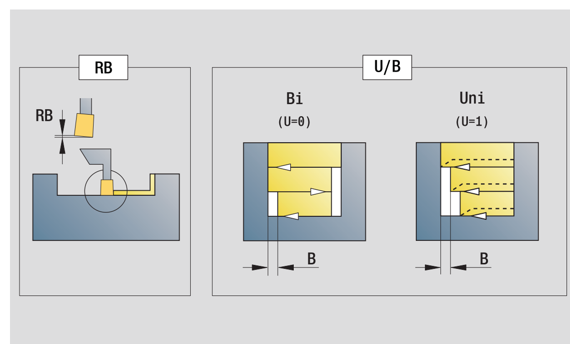
The Control uses the tool definition to distinguish between radial and axial recessing.

Turning depth compensation RB: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor. The value is usually determined empirically.

Offset width B: After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the offset width B. Each time the system switches on this side, the path is reduced by B—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius). If required, the Control reduces the programmed offset width. After clearance roughing, the remaining material is removed with a single cut.







Access to the technology database:

- Machining operation: Recess turning
- Affected parameters: F, S, O, P

"Parting" unit

The unit parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter. At the end of cycle, the tool returns to the starting point. You can define a feed rate reduction, which becomes effective as soon as the position **I** is reached.

Unit name: G859_CUT_OFF / Cycle: G859 (see page 305)

Cycle form

X1, Z1 Starting point of contour in X, Z (X: diameter value)

B Chamfer/rounding

- B>0: Radius of rounding
- B<0: Section length of chamfer

XE Inside diameter (pipe)

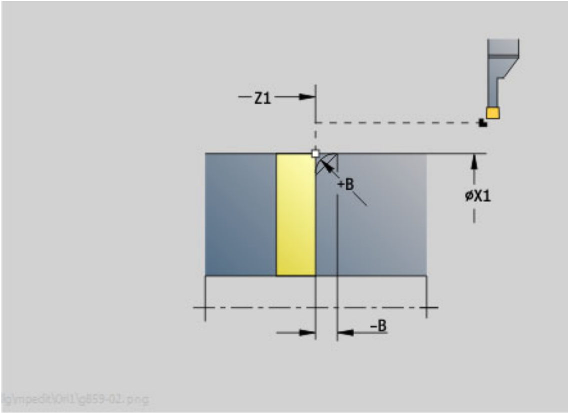
I Diameter for feed-rate reduction. Limit diameter over which traverse is at reduced feed rate.

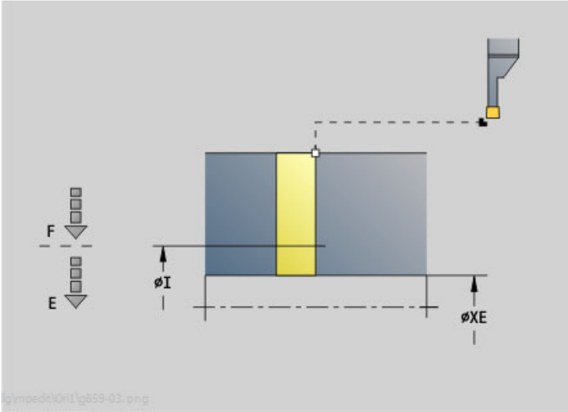
E Reduced feed rate

D Maximum speed

Further forms: see page 58

The limit to the maximum speed **"D"** is only effective in the cycle. After the cycle ends, the speed limit before the cycle becomes effective.





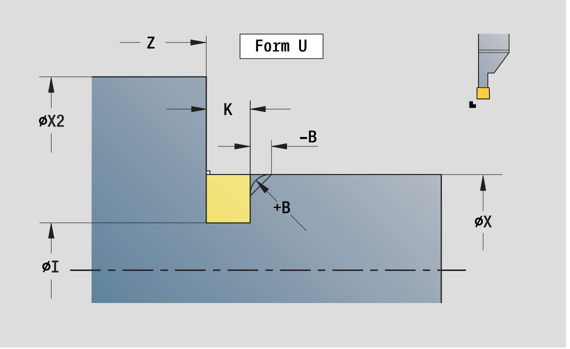
Access to the technology database:

- Machining operation: Contour recessing
- Affected parameters: F, S, E

"Undercutting (H, K, U)" unit

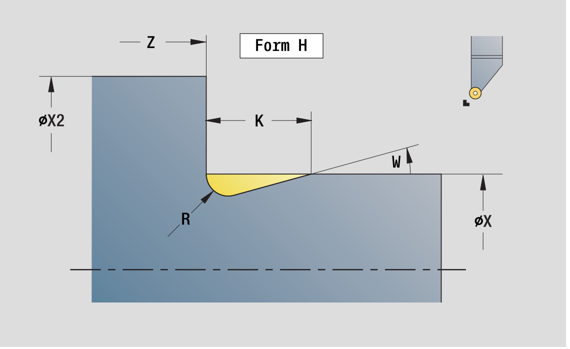
Depending on **KG**, the unit machines one of the following undercuts:

- Form U: The unit machines an undercut and finishes the adjoining plane surface. Either a chamfer or a rounding arc can be machined.
- Form H: The end point of the undercut is determined from the plunging angle.
- Form K. Only one linear cut at an angle of 45° is performed. The resulting contour geometry therefore depends on the tool that is used.



- First, you select the **Type of undercut KG**, and then you enter the values for the selected type of undercut.
- The Control changes parameters with the same address letters for the other undercuts as well. Do not change these values.

Unit name: G85x_H_K_U / Cycle: G85 (see page 306)



Contour form

KG Type of undercut

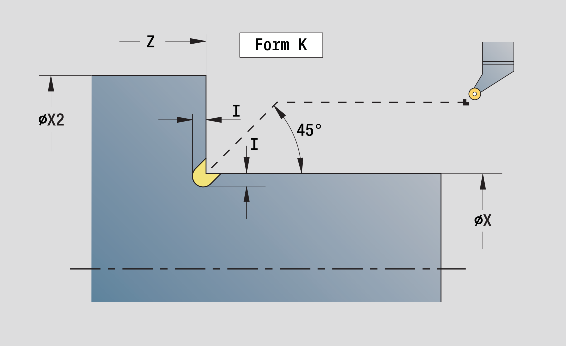
- Form U: Cycle G856 (see page 311)
- Form H: Cycle G857 (see page 312)
- Form K: Cycle G858 (see page 313)

X1, Z1 Contour corner point (X: diameter value)

Undercut type U

X2 End point, face (diameter value)
I Undercut diameter
K Undercut length
B Chamfer/rounding

- B>0: Radius of rounding
- B<0: Section length of chamfer



Undercut type H

K Undercut length
R Radius in the undercut corner
W Plunging angle

Undercut type K

I Undercut depth (radius)

Further forms: see page 58

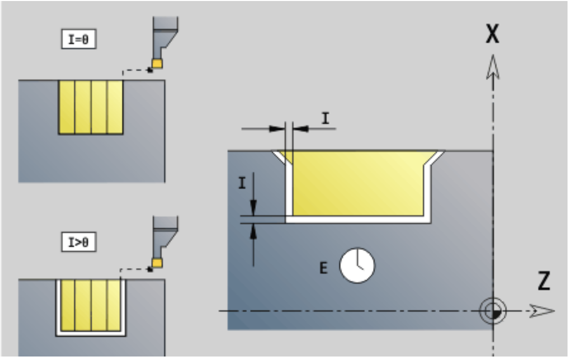
Access to the technology database:

- Machining operation: Finishing
- Affected parameters: F, S

"ICP recessing" unit

G870 generates a recess defined by G22-Geo. The Control uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Unit name: G870_ICP / Cycle: G870 (see page 281)



Contour form

I	Oversize in X and Z direction
EZ	Period of dwell after recessing path (default: time for one spindle revolution)

Further parameters of the contour form: see page 60.

Further forms: see page 58

Access to the technology database:

- Machining operation: Recessing
- Affected parameters: F, S

2.4 Units—Centric drilling

"Centric drilling" unit

The unit uses stationary tools to drill axial holes in several passes. Suitable tools can be positioned up to +/– 2 mm outside the turning center.

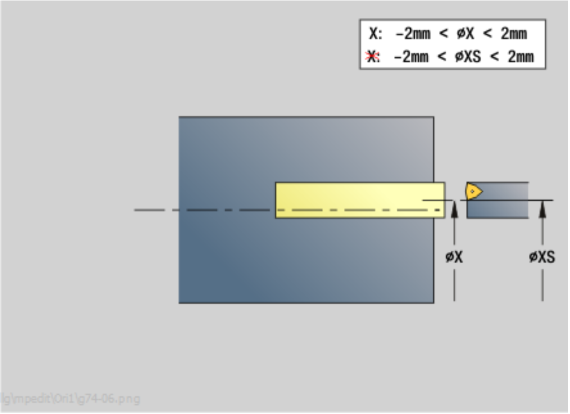
Unit name: G74_ZENTR / Cycle: G74 (see page 321)

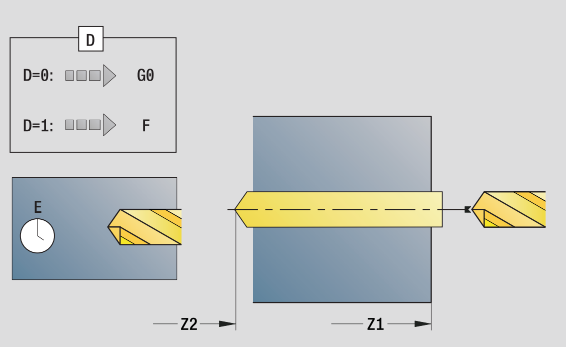
Cycle form

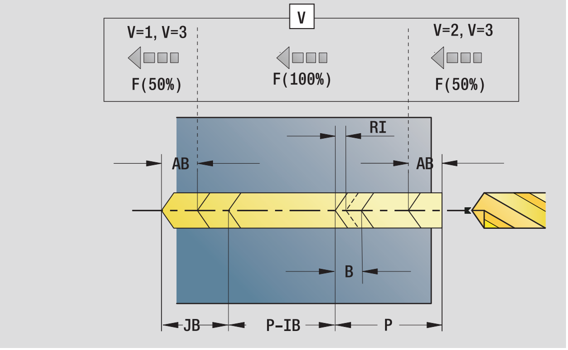
Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
NS	Starting block no. of contour
X	Start point drill (starting point of hole; diameter value)—(range: –2 mm < X < 2 mm; default: 0)
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
V	Feed rate reduction ■ 0: Without reduction ■ 1: At end of the hole ■ 2: At start of the hole ■ 3: At start and end of the hole
AB	Spot drilling / through drilling length (distance for feed rate reduction)
P	Hole depth
IB	Hole depth reduction value: Value by which the feed depth decreases after every advance.
JB	Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
B	Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.
RI	Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).

"Global" form

G14	Tool change point ■ No axis ■ 0: Simultaneously ■ 1: First X, then Z ■ 2: First Z, then X ■ 3: Only X ■ 4: Only Z ■ 5: Y direction only ■ 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)







Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

CLT	Coolant ■ 0: Without ■ 1: Circuit 1 on ■ 2: Circuit 2 on
SCK	Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.
G60	Protection zone. During drilling and boring the protection zone monitoring is ■ 0: Active ■ 1: Inactive
BP	Break duration: Time span for interruption of the feed for chip breaking.
BF	Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

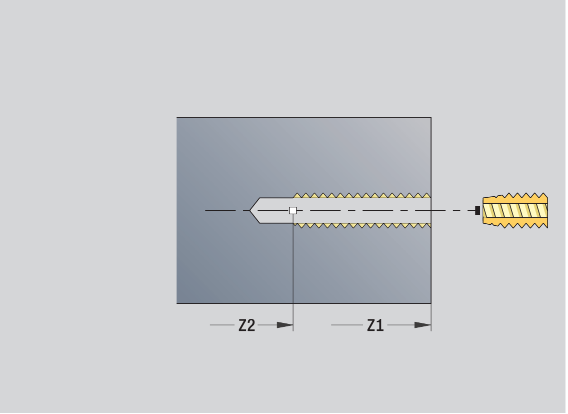
Further forms: see page 58

If **X** is not programmed or **XS** is in the range of –2 mm < XS < 2 mm, then the MANUALplus drills at **XS**.

"Centric tapping" unit

The unit cuts axial threads using stationary tools.

Unit name: G73_CENTR / Cycle: G73 (see page 318)

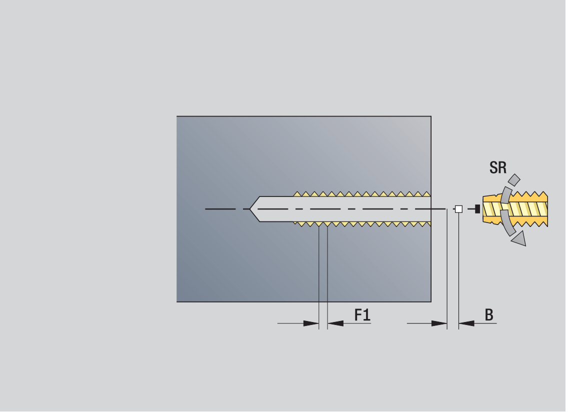


Cycle form

Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
NS	Starting block no. of contour
X	Start point drill (starting point of hole; diameter value)—(range: –2 mm < X < 2 mm; default: 0)
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)

Further forms: see page 58

Retraction length L: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retraction length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from the taps.



Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"Boring, centric countersinking" unit

The unit uses stationary tools to drill axial holes in several passes.

Unit name: G72_CENTR / Cycle: G72 (see page 317)

Cycle form

NS Starting block no. of contour

E Delay (dwell time at end of hole) (default: 0)

D Retraction at

- 0: Rapid traverse
- 1: Feed rate

RB Return plane

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

G60 Protection zone. During drilling and boring the protection zone monitoring is

- 0: Active
- 1: Inactive

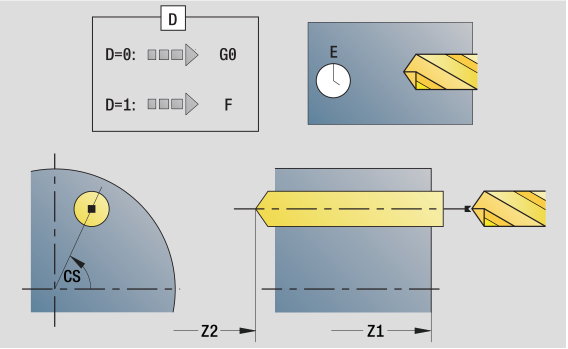
Further forms: see page 58

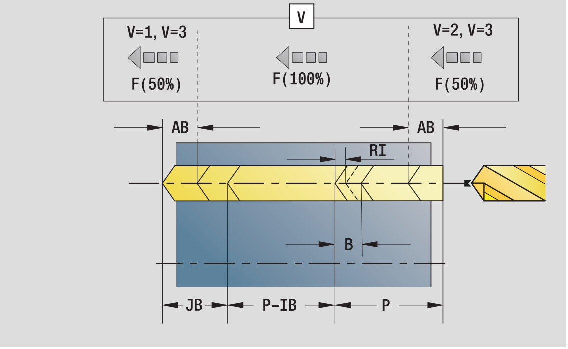
2.5 Units—Drilling in C axis

"Single hole, face" unit

This unit machines a hole on the face of the workpiece.

Unit name: G74_Bohr_Stirn_C / Cycle: G74 (see page 321)





Cycle form

Z1 Start point drill (starting point of hole)
Z2 End point drill (end point of hole)
CS Spindle angle
E Delay (dwell time at end of hole) (default: 0)
D Retraction at

- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction

- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length – distance for feed rate reduction
P Hole depth
IB Hole depth reduction value: Value by which the feed depth decreases after every advance.
JB Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
B Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.
RI Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

CLT	Coolant ■ 0: Without ■ 1: Circuit 1 on ■ 2: Circuit 2 on
SCK	Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.
G60	Protection zone. During drilling and boring the protection zone monitoring is ■ 0: Active ■ 1: Inactive
BP	Break duration: Time span for interruption of the feed for chip breaking.
BF	Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

Further forms: see page 58

"Linear pattern drilling, face" unit

The unit machines a linear drilling pattern in which the individual features are arranged at a regular spacing on the face.

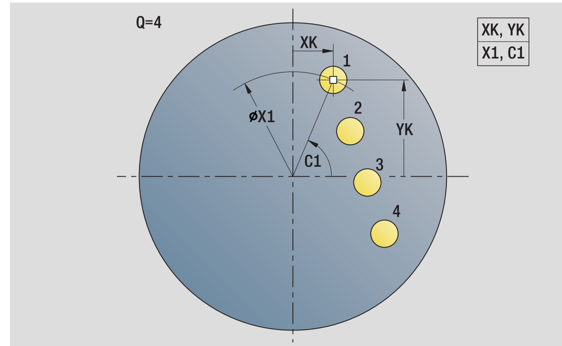
Unit name: G74_Lin_Stirn_C / Cycle: G74 (see page 321)

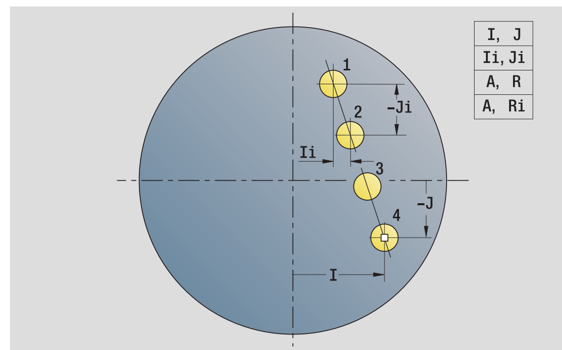
Pattern form

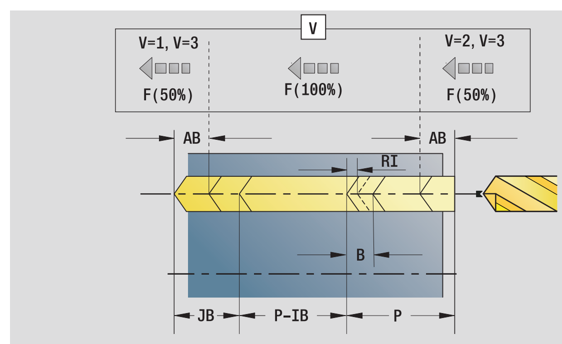
Q	Number of holes
X1, C1	Polar starting point
XK, YK	Cartesian starting point
I, J	End point (XK, YK)
Ii, Ji:	Distance (XKi, YKi)
R	Distance to first/last hole
Ri	Incremental distance
A	Pattern angle (reference is XK axis)

Cycle form

Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
V	Feed rate reduction ■ 0: Without reduction ■ 1: At end of the hole ■ 2: At start of the hole ■ 3: At start and end of the hole
AB	Spot drilling / through drilling length (distance for feed rate reduction)
P	Hole depth
IB	Hole depth reduction value: Value by which the feed depth decreases after every advance.
JB	Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
B	Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.
RI	Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).
RB	Return plane (default: return to the starting position or to the safety clearance)







Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

G60 Protection zone. During drilling and boring the protection zone monitoring is

- 0: Active
- 1: Inactive

BP Break duration: Time span for interruption of the feed for chip breaking.

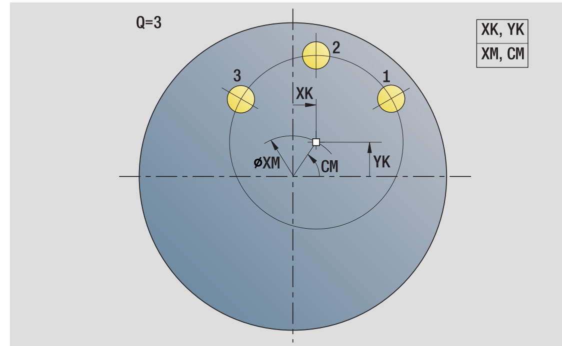
BF Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

Further forms: see page 58

"Circular pattern drilling, face" unit

This unit machines a circular drilling pattern on the face of the workpiece.

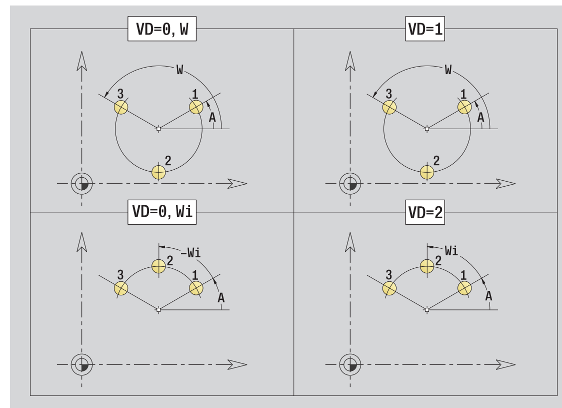
Unit name: G74_Cir_Stirn_C / Cycle: G74 (see page 321)



Pattern form

Q	Number of holes
XM, CM	Polar center point
XK, YK	Cartesian center point
A	Starting angle
Wi	Angle increment
K	Pattern diameter
W	End angle
VD	Rotation direction (default: 0)

- VD=0, without W: Figures are arranged on a full circle
- VD=0, with W: Figures are arranged on the longer circular arc
- VD=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- VD=1, with W: Clockwise
- VD=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- VD=2, with W: Counterclockwise
- VD=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)



Cycle form

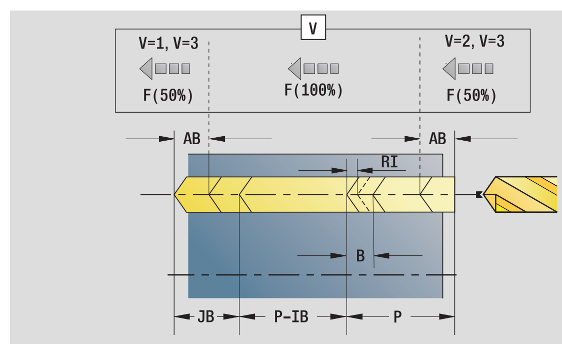
Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at

- 0: Rapid traverse
- 1: Feed rate

V	Feed rate reduction

- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB	Spot drilling / through drilling length (distance for feed rate reduction)
P	First hole depth
IB	Hole depth reduction value: Value by which the feed depth decreases after every advance.
JB	Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
B	Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.



Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

RI Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).

RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

G60 Protection zone. During drilling and boring the protection zone monitoring is

- 0: Active
- 1: Inactive

BP Break duration: Time span for interruption of the feed for chip breaking.

BF Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

Further forms: see page 58

"Tapping, face" unit

This unit machines a single tap hole on the face of the workpiece.

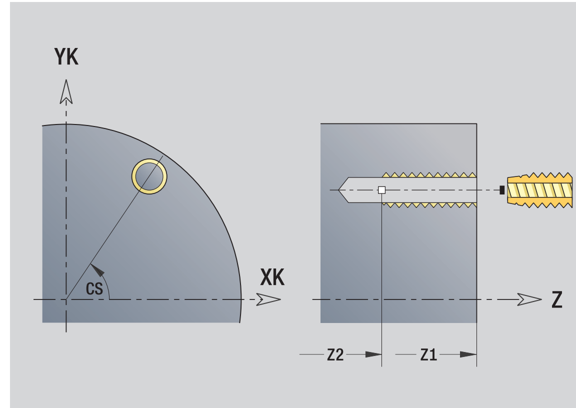
Unit name: G73_Gew_Stirn_C / Cycle: G73 (see page 318)

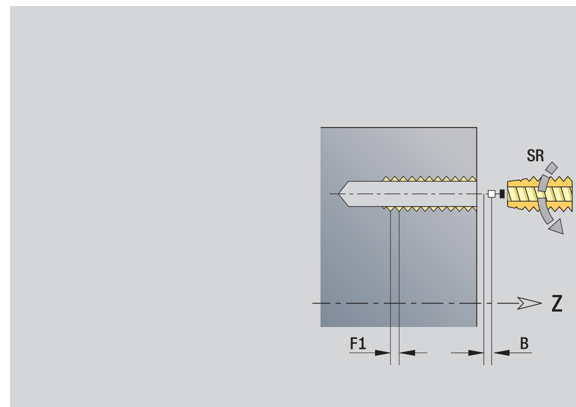
Cycle form

Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
CS	Spindle angle
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)

Further forms: see page 58

Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.





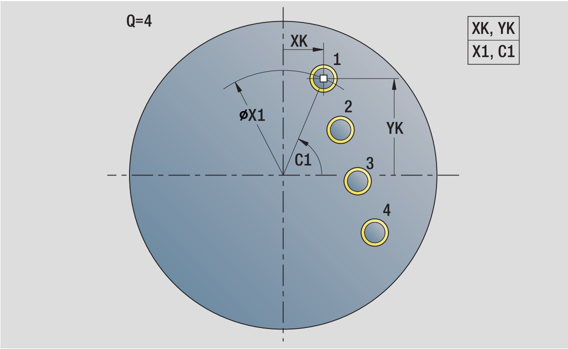
Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"Linear tapping pattern, face" unit

The unit machines a linear tapping pattern in which the individual features are arranged at a regular spacing on the face.

Unit name: G73_Lin_Stirn_C / Cycle: G73 (see page 318)



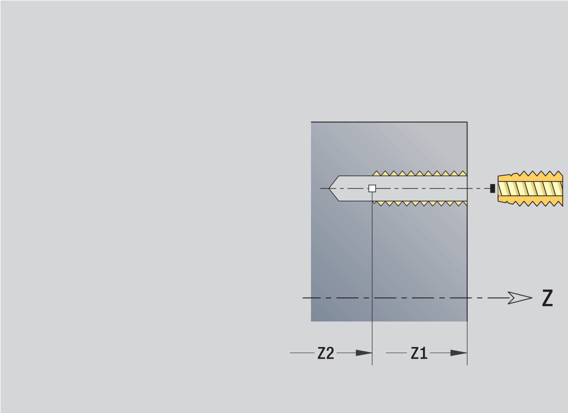
Pattern form

Q	Number of holes
X1, C1	Polar starting point
XK, YK	Cartesian starting point
I, J	End point (XK, YK)
Ii, Ji:	Distance (XKi, YKi)
R	Distance to first/last hole
Ri	Incremental distance
A	Pattern angle (reference is XK axis)

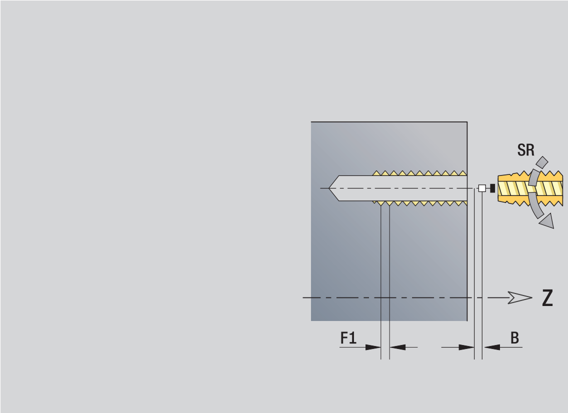
Cycle form

Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)
RB	Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58



Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.



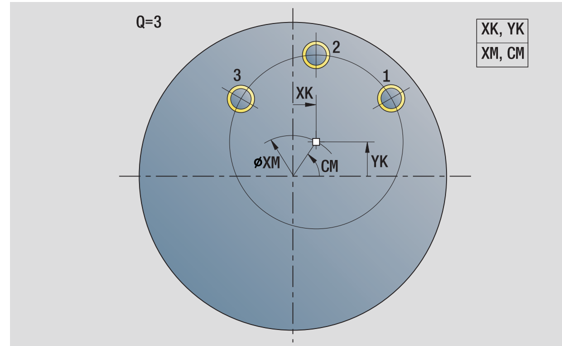
Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"Circular tapping pattern, face" unit

This unit machines a circular tapping pattern on the face of the workpiece.

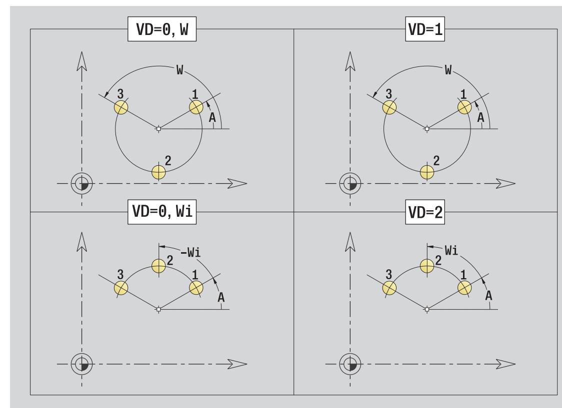
Unit name: G73_Cir_Stirn_C / Cycle: G73 (see page 318)



Pattern form

Q	Number of holes
XM, CM	Polar center point
XK, YK	Cartesian center point
A	Starting angle
Wi	Angle increment
K	Pattern diameter
W	End angle
VD	Rotation direction (default: 0)

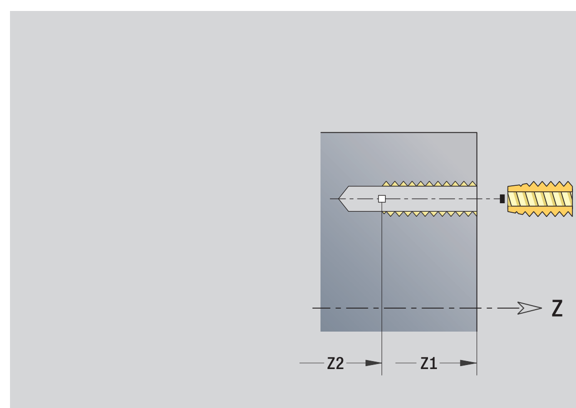
- VD=0, without W: Figures are arranged on a full circle
- VD=0, with W: Figures are arranged on the longer circular arc
- VD=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- VD=1, with W: Clockwise
- VD=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- VD=2, with W: Counterclockwise
- VD=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)



Cycle form

Z1	Start point drill (starting point of hole)
Z2	End point drill (end point of hole)
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)
RB	Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58



Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.

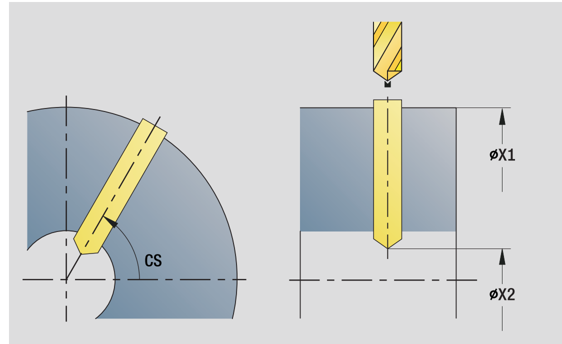
Access to the technology database:

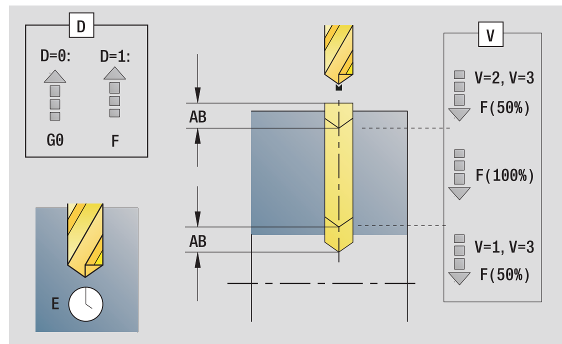
- Machining operation: Tapping
- Affected parameter: S

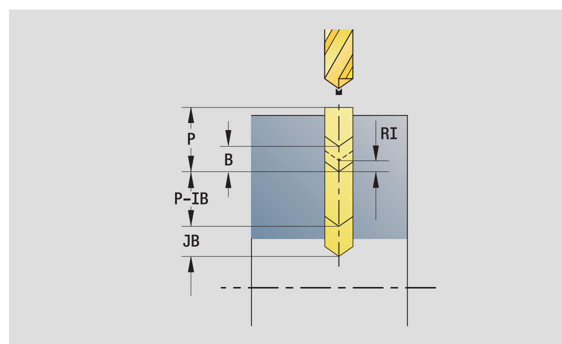
"Single hole, lateral surface" unit

This unit machines a hole on the lateral surface of the workpiece.

Unit name: G74_Bohr_Mant_C / Cycle: G74 (see page 321)







Cycle form

X1 Start point drill (starting point of hole; diameter value)

X2 End point drill (end point of hole; diameter value)

CS Spindle angle

E Delay (dwell time at end of hole) (default: 0)

D Retraction at

- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction

- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)

P Hole depth

IB Hole depth reduction value: Value by which the feed depth decreases after every advance.

JB Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.

B Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.

RI Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

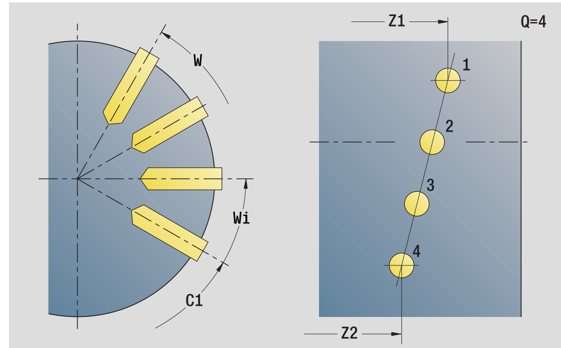
SCK	Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.
BP	Break duration: Time span for interruption of the feed for chip breaking.
BF	Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

Further forms: see page 58

"Linear pattern drilling, lateral surface" unit

The unit machines a linear drilling pattern in which the individual features are arranged at a regular spacing on the lateral surface.

Unit name: G74_Lin_Mant_C / Cycle: G74 (see page 321)

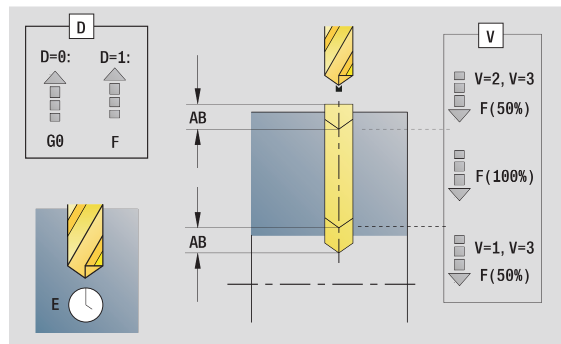


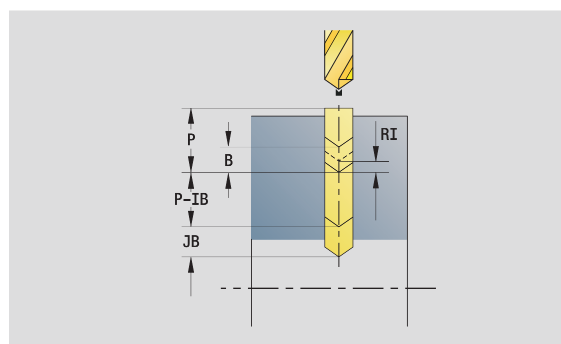
Pattern form

Q	Number of holes
Z1, C1	Starting point of pattern
Wi	Angle increment
W	End angle
Z2	End point of pattern

Cycle form

X1	Start point drill (starting point of hole; diameter value)
X2	End point drill (end point of hole; diameter value)
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
V	Feed rate reduction ■ 0: Without reduction ■ 1: At end of the hole ■ 2: At start of the hole ■ 3: At start and end of the hole
AB	Spot drilling / through drilling length (distance for feed rate reduction)
P	Hole depth
IB	Hole depth reduction value: Value by which the feed depth decreases after every advance.
JB	Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
B	Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.
RI	Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).
RB	Return plane (default: return to the starting position or to the safety clearance)





Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

BP Break duration: Time span for interruption of the feed for chip breaking.

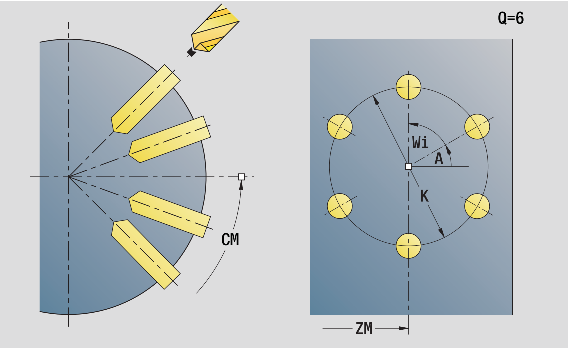
BF Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

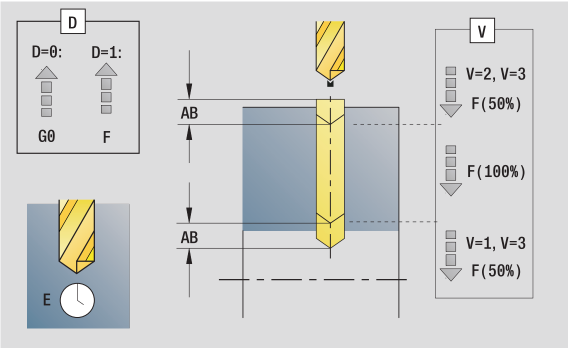
Further forms: see page 58

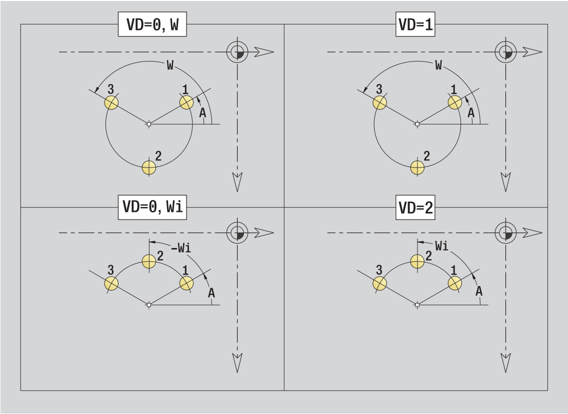
"Circular pattern drilling, lateral surface" unit

This unit machines a circular hole pattern on the lateral surface of the workpiece.

Unit name: G74_Cir_Mant_C / Cycle: G74 (see page 321)







Pattern form

Q	Number of holes
ZM, CM	Center point of pattern
A	Starting angle
Wi	Angle increment
K	Pattern diameter
W	End angle
VD	Rotation direction (default: 0)

- VD=0, without W: Figures are arranged on a full circle
- VD=0, with W: Figures are arranged on the longer circular arc
- VD=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- VD=1, with W: Clockwise
- VD=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- VD=2, with W: Counterclockwise
- VD=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)

Cycle form

X1	Start point drill (starting point of hole; diameter value)
X2	End point drill (end point of hole; diameter value)
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at:

- 0: Rapid traverse
- 1: Feed rate

V	Feed rate reduction:

- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB	Spot drilling / through drilling length (distance for feed rate reduction)
P	Hole depth
IB	Hole depth reduction value: Value by which the feed depth decreases after every advance.

Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

JB Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.

B Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.

RI Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).

RB Return plane (default: return to the starting position or to the safety clearance)

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

BP Break duration: Time span for interruption of the feed for chip breaking.

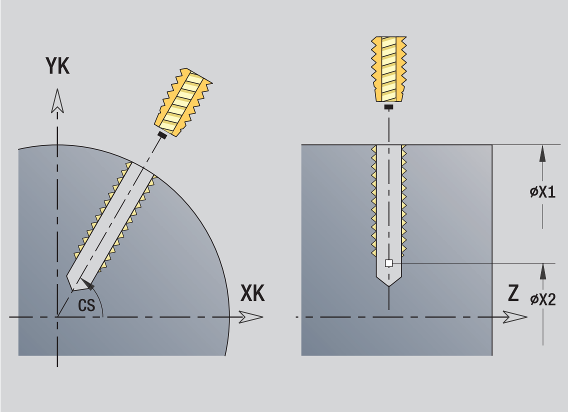
BF Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

Further forms: see page 58

"Tap hole, lateral surface" unit

This unit machines a tap hole on the lateral surface of the workpiece.

Unit name: G73_Gew_Mant_C / Cycle: G73 (see page 318)

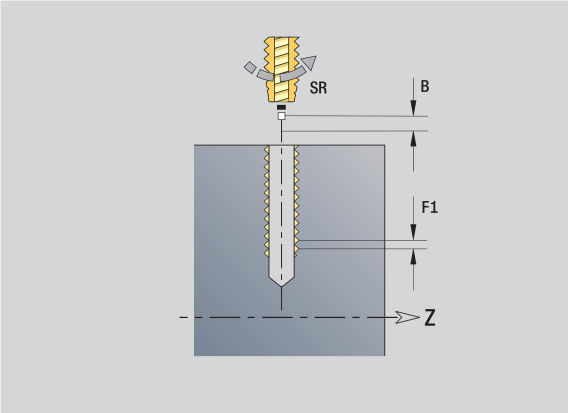


Cycle form

X1	Start point drill (starting point of hole; diameter value)
X2	End point drill (end point of hole; diameter value)
CS	Spindle angle
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)

Further forms: see page 58

Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.



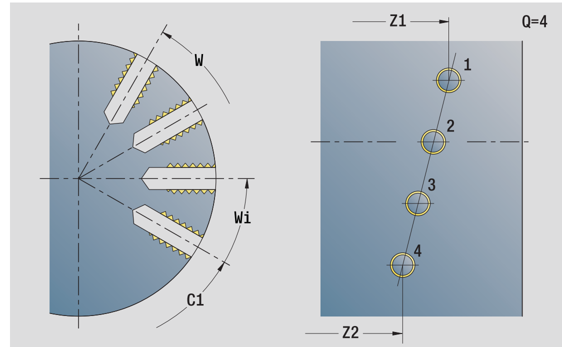
Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"Linear tapping pattern, lateral surface" unit

The unit machines a linear tapping pattern in which the individual features are arranged at a regular spacing on the lateral surface.

Unit name: G73_Lin_Mant_C / Cycle: G73 (see page 318)



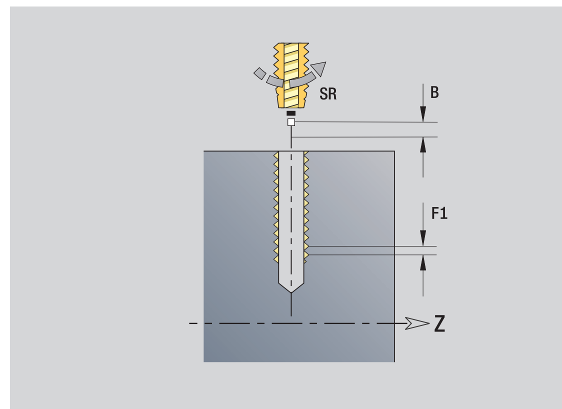
Pattern form

Q	Number of holes
Z1, C1	Starting point of pattern
Wi	Angle increment
W	End angle
Z2	End point of pattern

Cycle form

X1	Start point drill (starting point of hole; diameter value)
X2	End point drill (end point of hole; diameter value)
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)
RB	Return plane

Further forms: see page 58



Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.

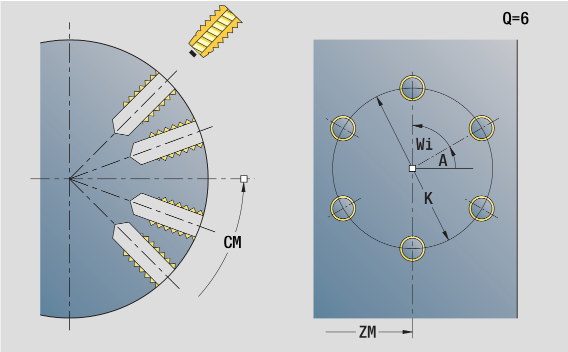
Access to the technology database:

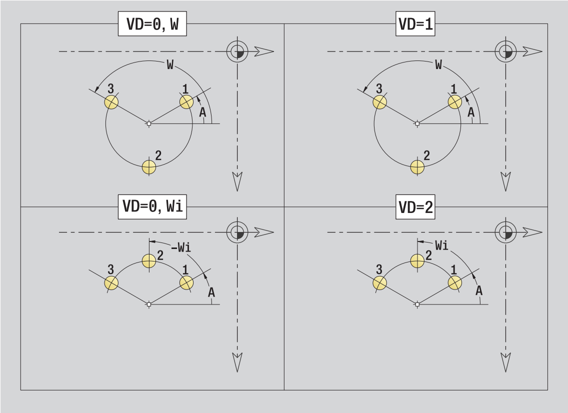
- Machining operation: Tapping
- Affected parameter: S

"Circular tapping pattern, lateral surface" unit

This unit machines a circular tapping pattern on the lateral surface of the workpiece.

Unit name: G73_Cir_Mant_C / Cycle: G73 (see page 318)





Pattern form

Q	Number of holes
ZM, CM	Center point of pattern
A	Starting angle
Wi	Angle increment
K	Pattern diameter
W	End angle
VD	Rotation direction (default: 0)

- VD=0, without W: Figures are arranged on a full circle
- VD=0, with W: Figures are arranged on the longer circular arc
- VD=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- VD=1, with W: Clockwise
- VD=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- VD=2, with W: Counterclockwise
- VD=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)

Cycle form

X1	Start point drill (starting point of hole; diameter value)
X2	End point drill (end point of hole; diameter value)
F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)
RB	Return plane

Further forms: see page 58

Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.

Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"ICP drilling, C axis" unit

The unit machines a single hole or a hole pattern on the face or lateral surface. Using ICP, you define the holes as well as further details.

Unit name: G74_ICP_C / Cycle: G74 (see page 321)

Pattern form

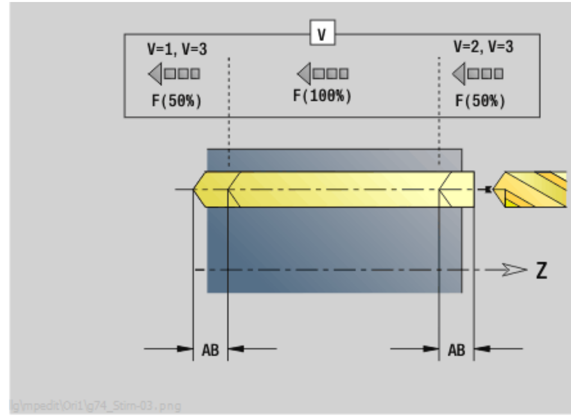
FK Finished part contour
NS Starting block no. of contour

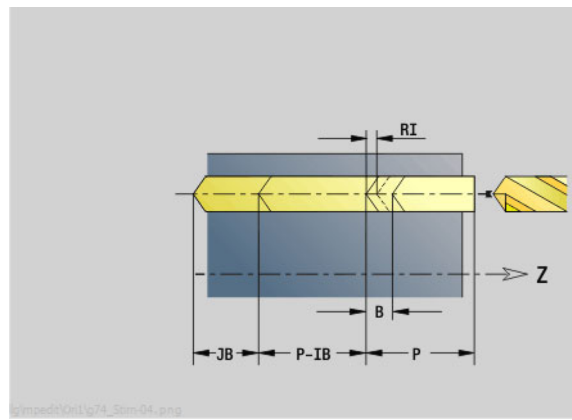
Cycle form

E Delay (dwell time at end of hole) (default: 0)
D Retraction at
- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)
P Hole depth
IB Hole depth reduction value: Value by which the feed depth decreases after every advance.
JB Minimum hole depth: If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
B Retraction distance: Value by which the tool is retracted after reaching the respective hole depth.
RI Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).
RB Return plane (default: return to the starting position or to the safety clearance)





Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

"Global" form

G14 Tool change point

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Y direction only
- 6: Simultaneous with Y (X, Y and Z axes move on a diagonal path)

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

SCK Safety clearance in infeed direction: Safety clearance in infeed direction during drilling and milling operations.

BP Break duration: Time span for interruption of the feed for chip breaking.

BF Feed duration: Time interval until the next break. The interruption of the feed rate breaks the chip.

Further forms: see page 58

"ICP tapping, C axis" unit

The unit machines a single tap hole or a tapping pattern on the face or lateral surface. Using ICP, you define the tap holes as well as further details.

Unit name: G73_ICP_C / Cycle: G73 (see page 318)

Pattern form

FK see page 60

NS Starting block no. of contour

Cycle form

F1 Thread pitch

B Run-in length

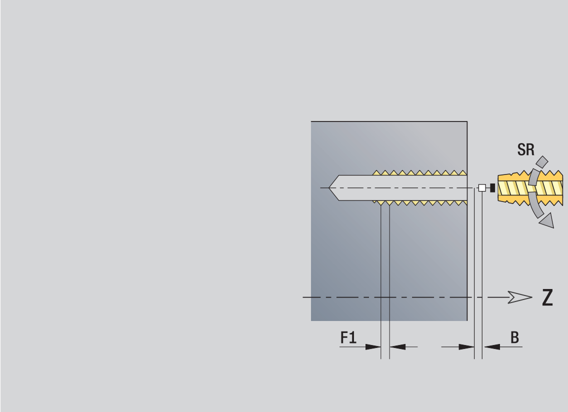
L Retraction length when using floating tap holders (default: 0)

SR Retraction speed (default: Shaft speed for tapping)

RB Return plane

Further forms: see page 58

Use the **retraction length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from taps.



Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"ICP boring/countersinking, C axis" unit

The unit machines a single hole or a hole pattern on the face or lateral surface. Using ICP, you define the hole positions as well as further details for boring or countersinking.

Unit name: G72_ICP_C / Cycle: G72 (see page 317)

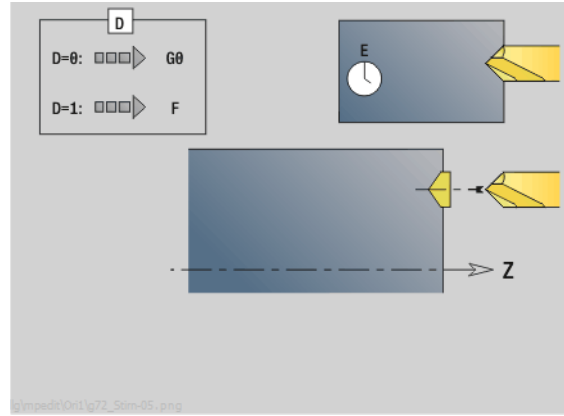
Pattern form

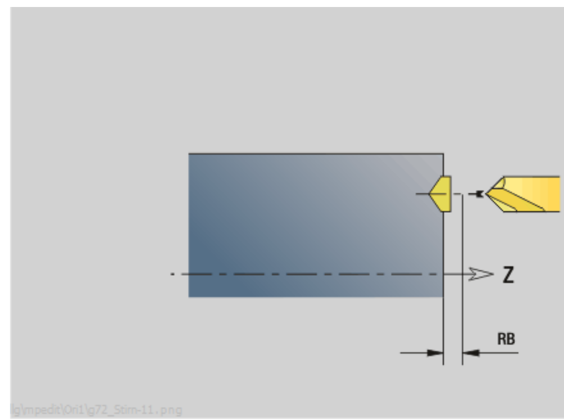
FK	see page 60
NS	Starting block no. of contour

Cycle form

E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
RB	Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58





Access to the technology database:

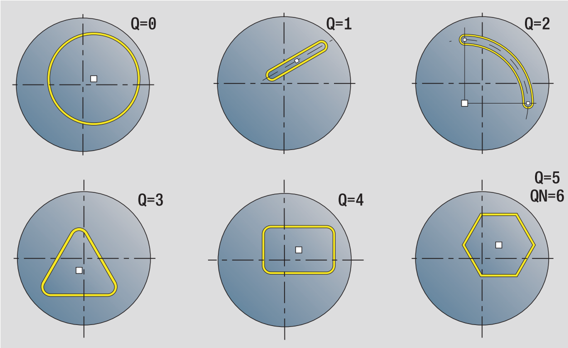
- Machining operation: Drilling
- Affected parameters: F, S

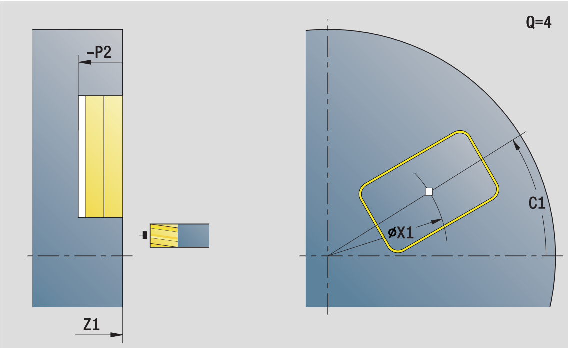
2.6 Units—Predrilling in C axis

"Predrill, contour mill, figures on face" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF.

Unit name: DRILL_STI_KON_C / Cycles: G840 A1 (see page 349); G71 (see page 315)





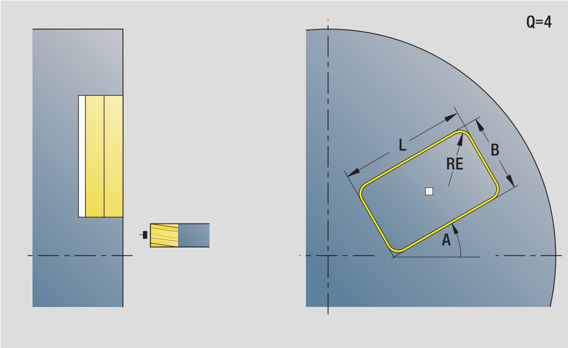


Figure form

Q Type of figure

- 0: Full circle
- 1: Linear slot
- 2: Circular slot
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

QN Number of polygon corners—only with Q=5 (polygon)
X1 Diameter of figure center
C1 Angle of figure center
Z1 Milling top edge
P2 Depth of figure
L Edge length / width across flats

- L>0: Edge length
- L<0: Width across flats (inside diameter) for polygon

B Rectangle width
RE Rounding radius
A Angle to X axis
Q2 Rotational direction of slot—only if Q=2 (circular slot)

- cw: In clockwise direction
- ccw: In counterclockwise direction

W Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.

Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

Cycle form

JK Cutter position

- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

I Contour-parallel oversize
K Infeed-direction oversize
R Approach radius
WB Cutter diameter
NF Position mark
E Delay (dwell time at end of hole) (default: 0)
D Retraction at

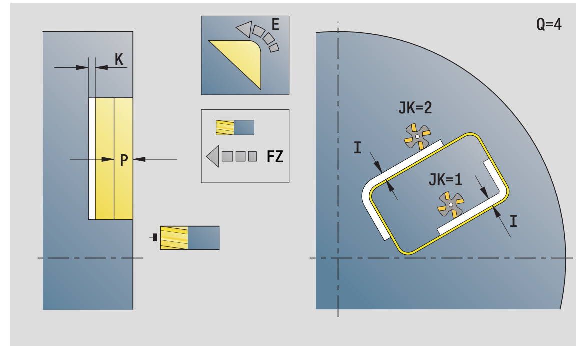
- 0: Rapid traverse
- 1: Feed rate

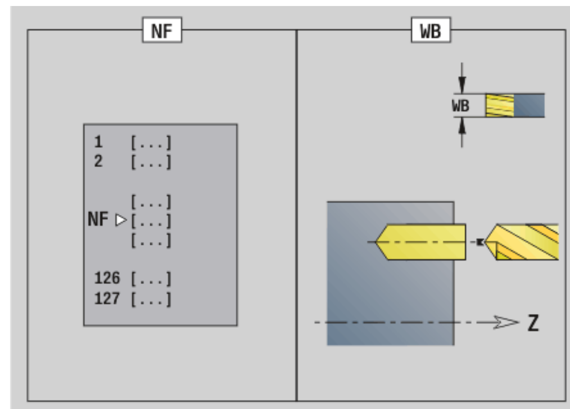
V Feed rate reduction

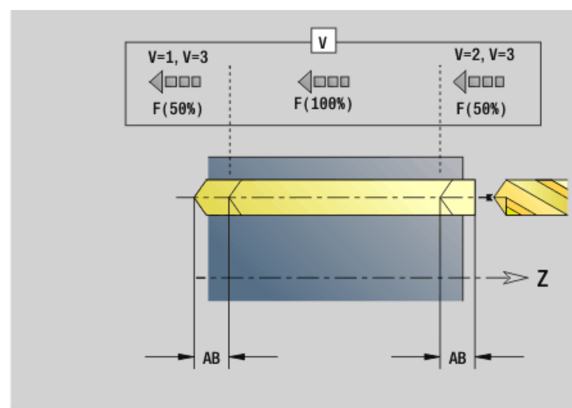
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)
RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58



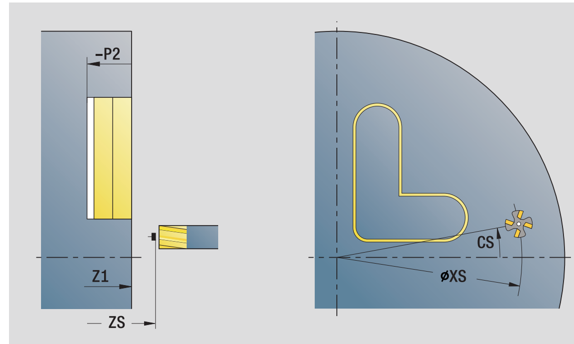




"Predrill, contour mill, ICP on face" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the milling contour consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_STI_840_C / Cycles: G840 A1 (see page 349); G71 (see page 315)



Contour form

FK see page 60
NS Starting block no. of contour
NE End block no. of contour
Z1 Milling top edge
P2 Depth of contour

Cycle form

JK Cutter position
- 0: On the contour
- 1, closed contour: Within the contour
- 1, open contour: Left of the contour
- 2, closed contour: Outside the contour
- 2, open contour: Right of the contour
- 3: Depending on H and MD

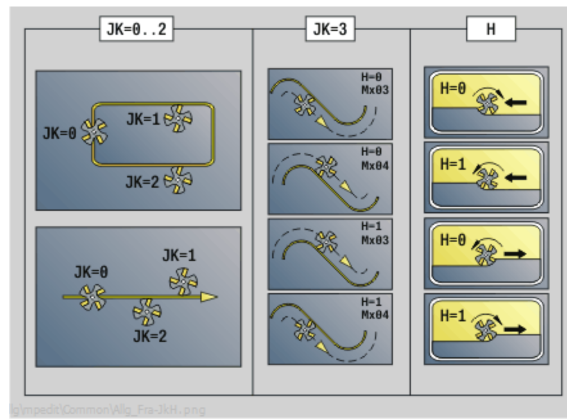
H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

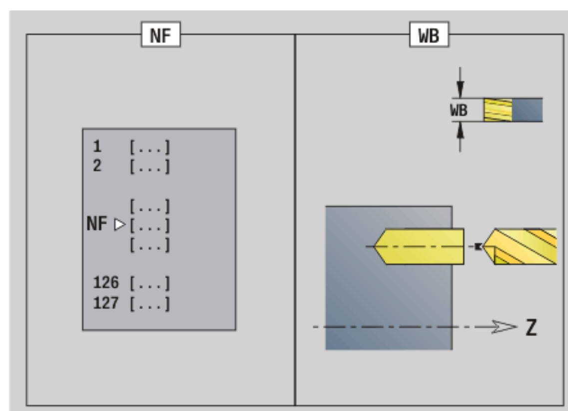
I Contour-parallel oversize
K Infeed-direction oversize
R Approach radius
WB Cutter diameter
NF Position mark
E Delay (dwell time at end of hole) (default: 0)
D Retraction at
- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)
RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58





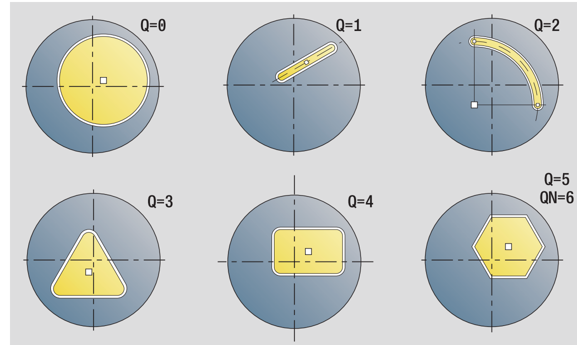
Access to the technology database:

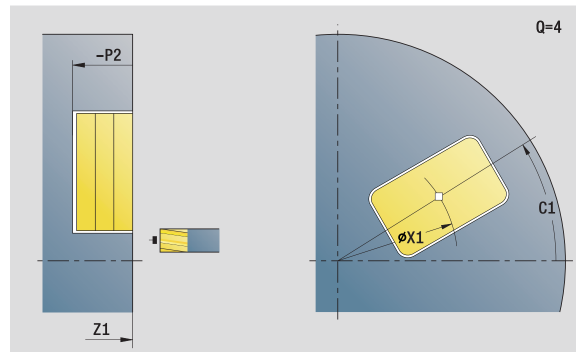
- Machining operation: Drilling
- Affected parameters: F, S

"Predrill, pocket mill, figures on face" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF.

Unit name: DRILL_STI_TASC / Cycles: G845 A1 (see page 359); G71 (see page 315)





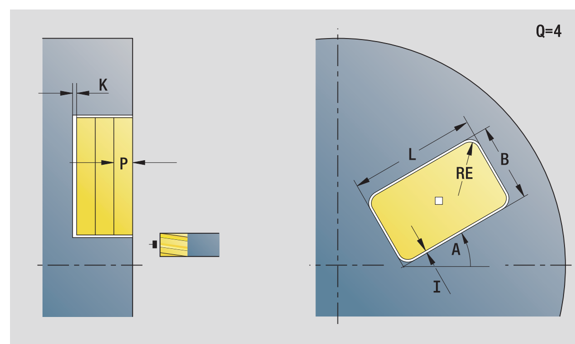


Figure form

Q Type of figure

- 0: Full circle
- 1: Linear slot
- 2: Circular slot
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

QN Number of polygon corners—only with Q=5 (polygon)

X1 Diameter of figure center

C1 Angle of figure center

Z1 Milling top edge

P2 Depth of figure

L Edge length / width across flats

- L>0: Edge length
- L<0: Width across flats (inside diameter) for polygon

B Rectangle width

RE Rounding radius

A Angle to X axis

Q2 Rotational direction of slot—only if Q=2 (circular slot)

- cw: In clockwise direction
- ccw: In counterclockwise direction

W Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.

Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

Cycle form

JT Machining direction

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

I Contour-parallel oversize

K Infeed-direction oversize

U Overlap factor (default: 0.5)

WB Cutter diameter

NF Position mark

E Delay (dwell time at end of hole) (default: 0)

D Retraction at

- 0: Rapid traverse
- 1: Feed rate

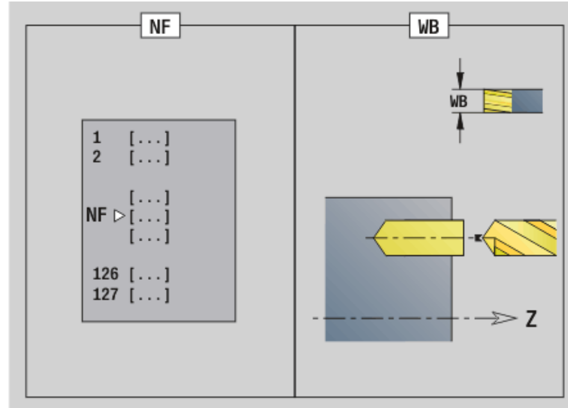
V Feed rate reduction

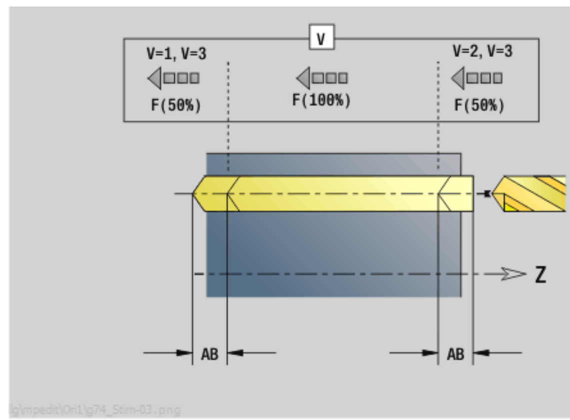
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)

RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58





"Predrill, pocket mill, ICP on face" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the pocket consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_STI_845_C / Cycles: G845 A1 (see page 359); G71 (see page 315)

Contour form

FK see page 60
NS Starting block no. of contour
NE End block no. of contour
Z1 Milling top edge
P2 Depth of contour

Cycle form

JT Machining direction
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

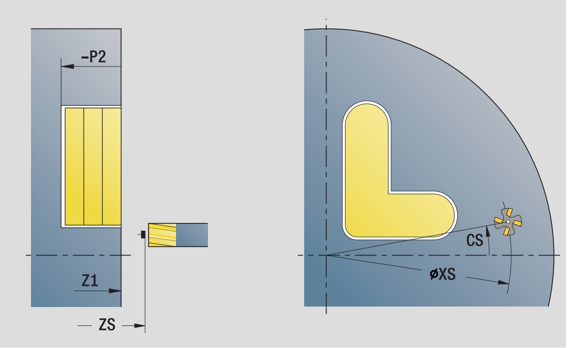
H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

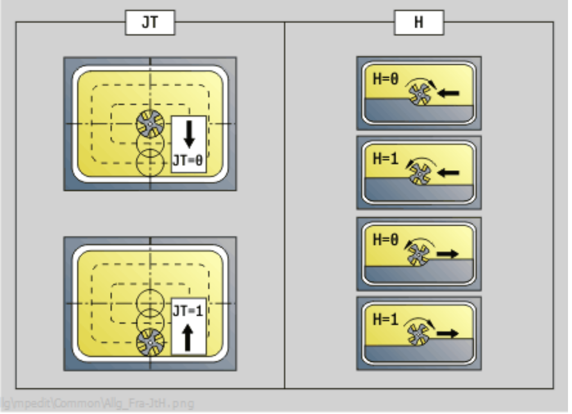
I Contour-parallel oversize
K Infeed-direction oversize
U Overlap factor (default: 0.5)
WB Cutter diameter
NF Position mark
E Delay (dwell time at end of hole) (default: 0)
D Retraction at
- 0: Rapid traverse
- 1: Feed rate

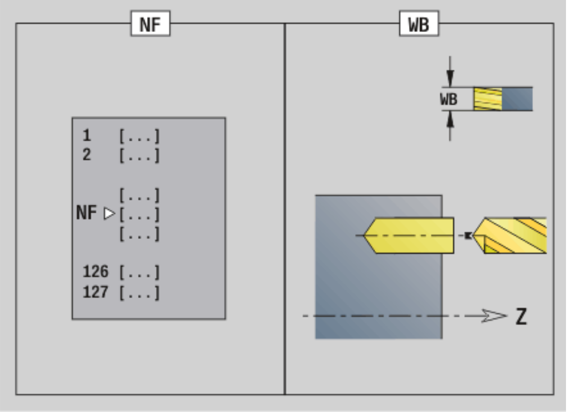
V Feed rate reduction
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)
RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58







Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

"Predrill, contour mill, figures on lateral surface" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF.

Unit name: DRILL_MAN_KON_C / Cycles: G840 A1 (see page 349); G71 (see page 315)

Figure form

Q Type of figure

- 0: Full circle
- 1: Linear slot
- 2: Circular slot
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

QN Number of polygon corners—only with Q=5 (polygon)
Z1 Figure center
C1 Angle of figure center
CY Figure center of unrolled lateral surface
X1 Milling top edge
P2 Depth of figure
L Edge length / width across flats

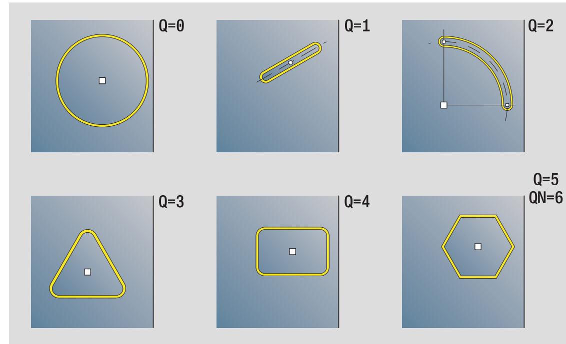
- L>0: Edge length
- L<0: Width across flats (inside diameter) for polygon

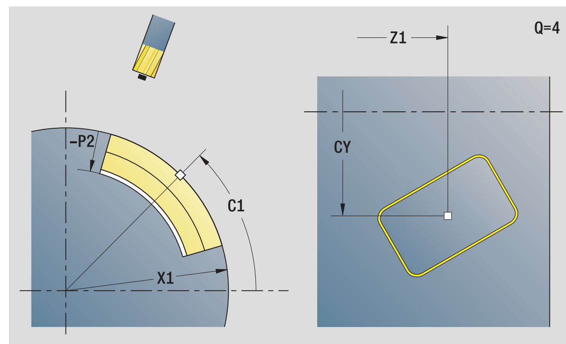
B Rectangle width
RE Rounding radius
A Angle to Z axis
Q2 Rotational direction of slot—only if Q=2 (circular slot)

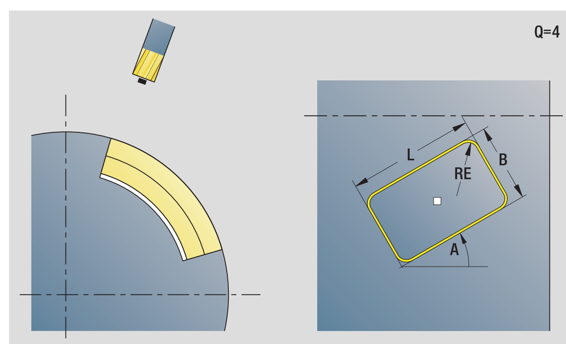
- cw: In clockwise direction
- ccw: In counterclockwise direction

W Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.







Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

Cycle form

JK Cutter position

- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

I Contour-parallel oversize

K Infeed-direction oversize

R Approach radius

WB Cutter diameter

NF Position mark

E Delay (dwell time at end of hole) (default: 0)

D Retraction at

- 0: Rapid traverse
- 1: Feed rate

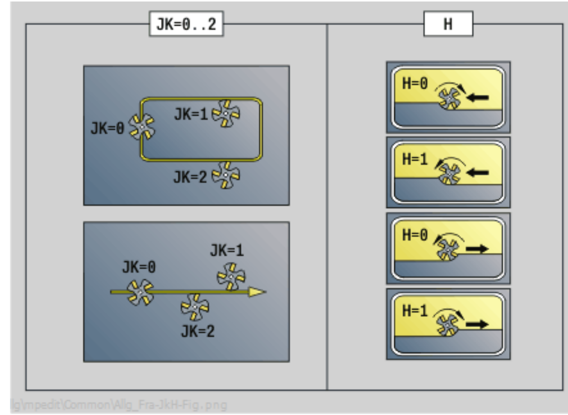
V Feed rate reduction

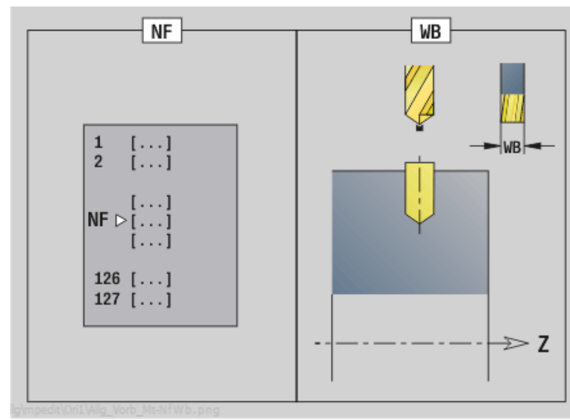
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

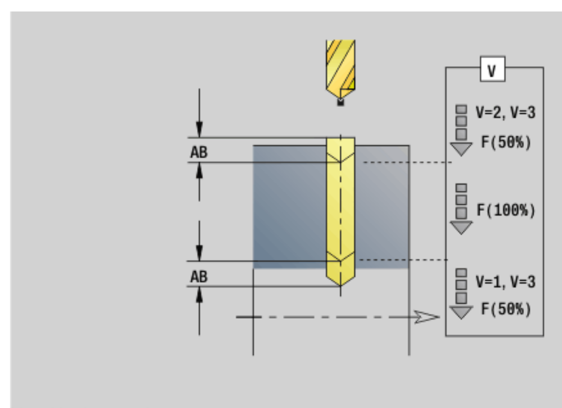
AB Spot drilling / through drilling length (distance for feed rate reduction)

RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58







"Predrill, contour mill, ICP on lateral surface" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the milling contour consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_MAN_840_C / Cycles: G840 A1 (see page 349); G71 (see page 315)

Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)
P2	Depth of contour (radius value)

Cycle form

JK Cutter position
- 0: On the contour
- 1, closed contour: Within the contour
- 1, open contour: Left of the contour
- 2, closed contour: Outside the contour
- 2, open contour: Right of the contour
- 3: Depending on H and MD

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

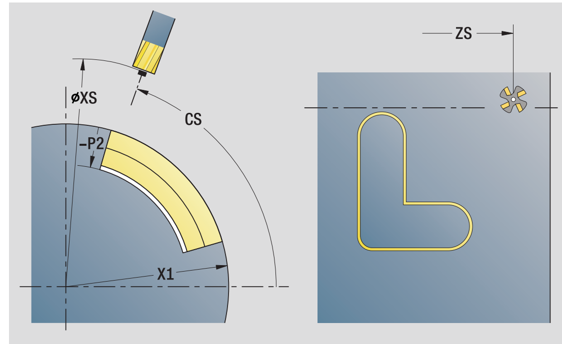
I	Contour-parallel oversize
K	Infeed-direction oversize
R	Approach radius
WB	Cutter diameter
NF	Position mark
E	Delay (dwell time at end of hole) (default: 0)

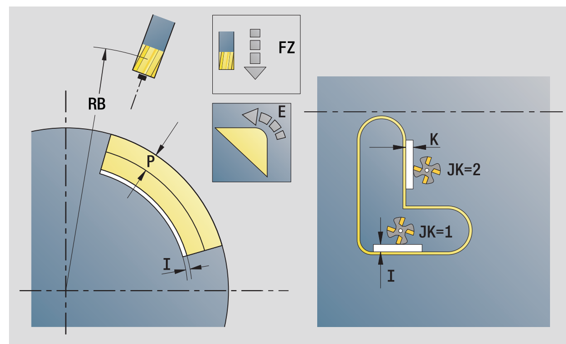
D Retraction at
- 0: Rapid traverse
- 1: Feed rate

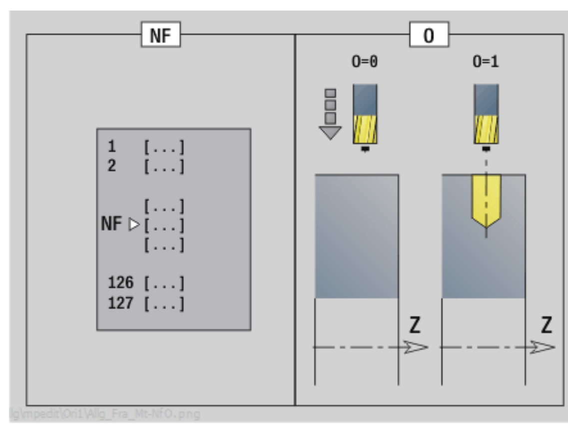
V Feed rate reduction
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB	Spot drilling / through drilling length (distance for feed rate reduction)
RB	Return plane (diameter value)

Further forms: see page 58







Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

"Predrill, pocket mill, figures on lateral surface" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF.

Unit name: DRILL_MAN_TAS_C / Cycles: G845 A1 (see page 359); G71 (see page 315)

Figure form

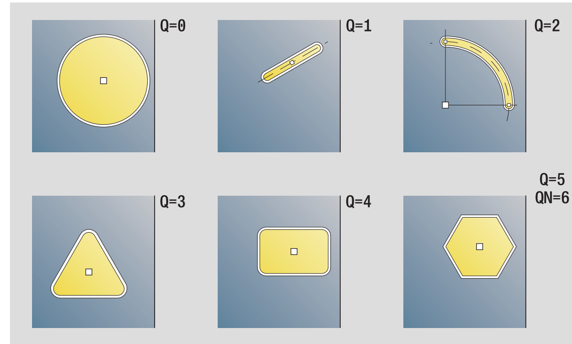
Q   Type of figure
- 0: Full circle
- 1: Linear slot
- 2: Circular slot
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

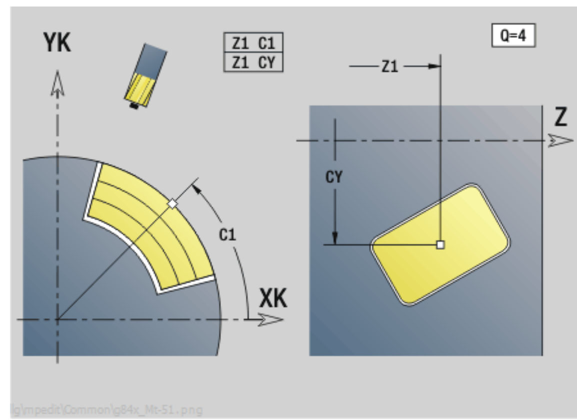
QN   Number of polygon corners—only with Q=5 (polygon)
Z1   Figure center
C1   Angle of figure center
CY   Figure center of unrolled lateral surface
X1   Milling top edge
P2   Depth of figure
L   Edge length / width across flats
- L>0: Edge length
- L<0: Width across flats (inside diameter) for polygon

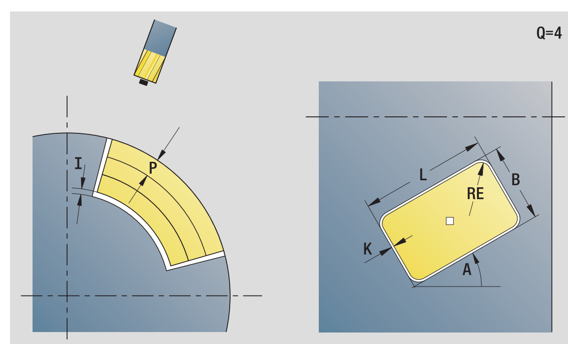
B   Rectangle width
RE   Rounding radius
A   Angle to Z axis
Q2   Rotational direction of slot—only if Q=2 (circular slot)
- cw: In clockwise direction
- ccw: In counterclockwise direction

W   Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.







Access to the technology database:
- Machining operation: Drilling
- Affected parameters: F, S

Cycle form

JT Machining direction

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

I Infeed-direction oversize

K Contour-parallel oversize

U Overlap factor (default: 0.5)

WB Cutter diameter

NF Position mark

E Delay (dwell time at end of hole) (default: 0)

D Retraction at

- 0: Rapid traverse
- 1: Feed rate

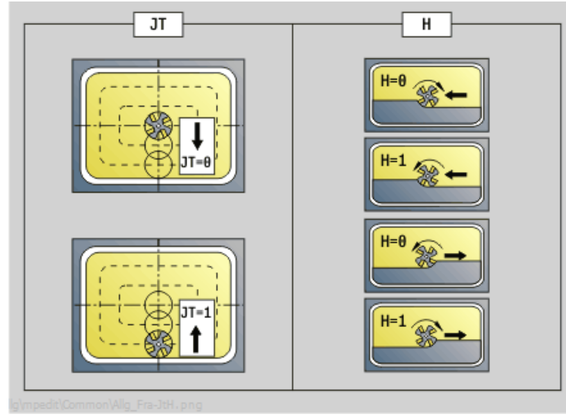
V Feed rate reduction

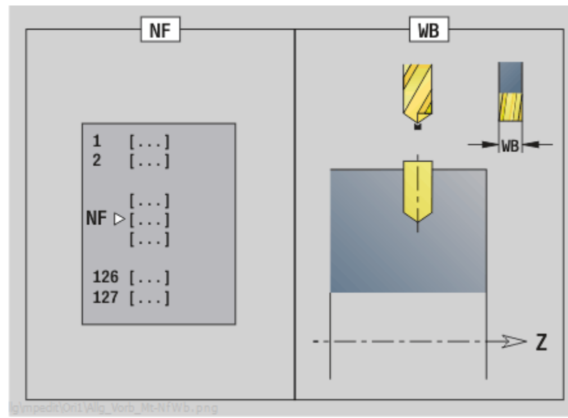
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

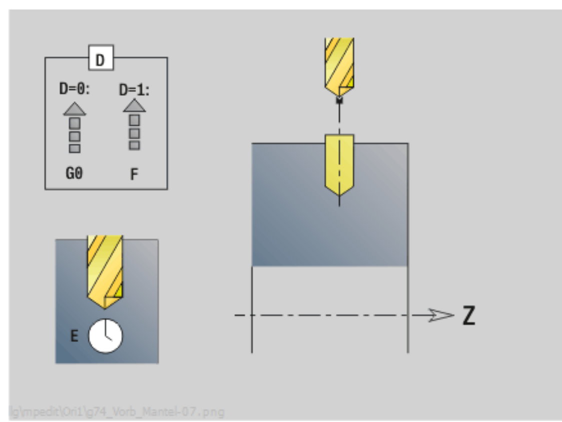
AB Spot drilling / through drilling length (distance for feed rate reduction)

RB Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58



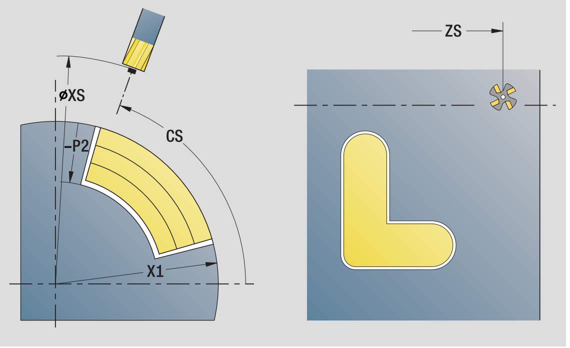




"Predrill, pocket mill, ICP on lateral surface" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the pocket consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_MAN_845_C / Cycles: G845 A1 (see page 359); G71 (see page 315)



Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)
P2	Depth of contour

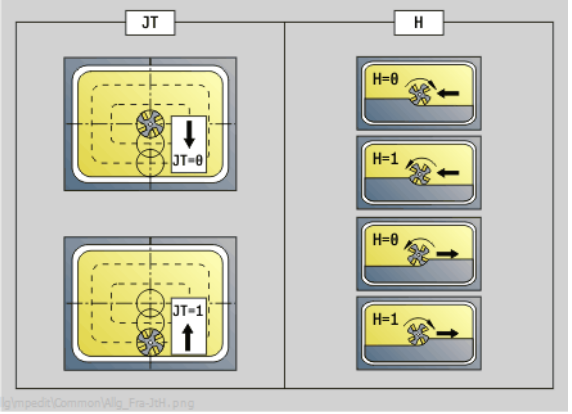
Cycle form

JT Machining direction

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

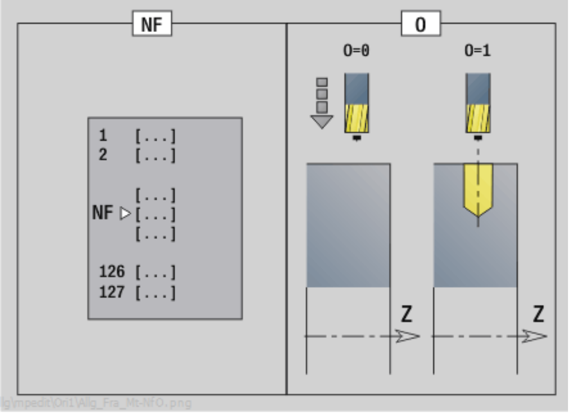


I	Infeed-direction oversize
K	Contour-parallel oversize
U	Overlap factor (default: 0.5)
WB	Cutter diameter
NF	Position mark
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at

- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction

- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole



AB	Spot drilling / through drilling length (distance for feed rate reduction)
RB	Return plane (diameter value)

Further forms: see page 58

Access to the technology database:

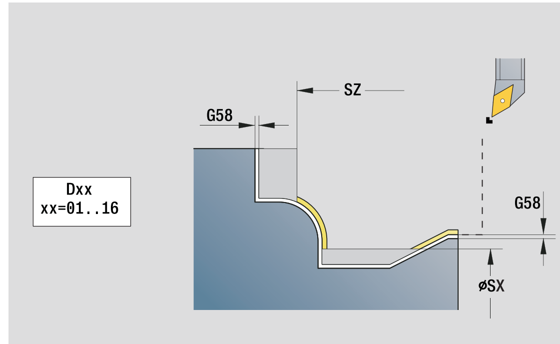
- Machining operation: Drilling
- Affected parameters: F, S

2.7 Units—Finishing

"ICP contour finishing" unit

The unit finishes the contour described by ICP from "NS to NE" in one pass.

Unit name: G890_ICP / Cycle: G890 (see page 282)



Contour form

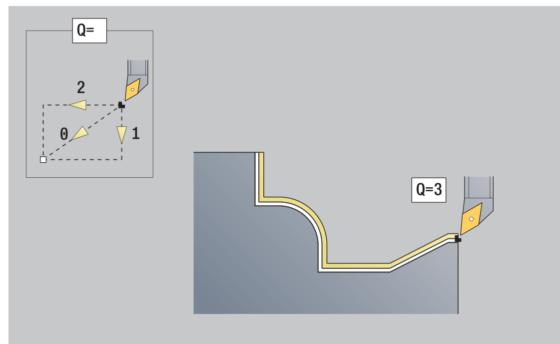
B Switch on TRC (type of tool radius compensation)

- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

SX, SZ Cutting limit (SX: diameter value)—(default: no cutting limit)

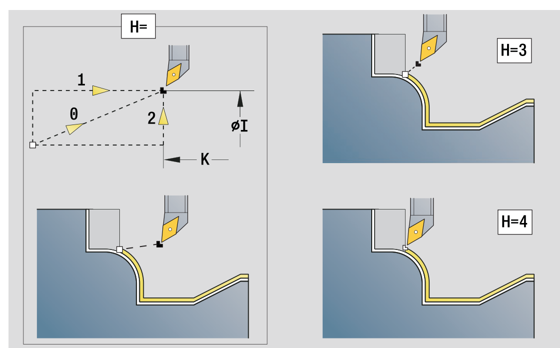
Further parameters of the contour form: see page 60.

Cycle form



Q Type of approach (default: 0)

- 0: Automatic selection—the Control checks:
  - Diagonal approach
  - First X, then Z direction
  - Equidistant around the barrier
  - Omission of the first contour elements if the starting position is inaccessible
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: No approach—tool is located near the starting point of the contour area.



H Type of retraction. Tool backs off at 45° against the machining direction and moves to the position I, K (default: 3):

- 0: Diagonal
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: Stops at safety clearance
- 4: No retraction motion (tool remains on the end coordinate)

I, K Cycle end position Position that is approached at the end of the cycle (I diameter value).

Access to the technology database:

- Machining operation: Finishing
- Affected parameters: F, S

Cycle form

D Omit elements (see figure)

E Plunging behavior

- E=0: Descending contours are not machined
- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

O Feed rate reduction for circular elements (default: 0)

- 0: Feed rate reduction is active
- 1: No feed rate reduction

DXX Additive correction numbers 1 – 16

G58 Contour-parallel oversize (radius)

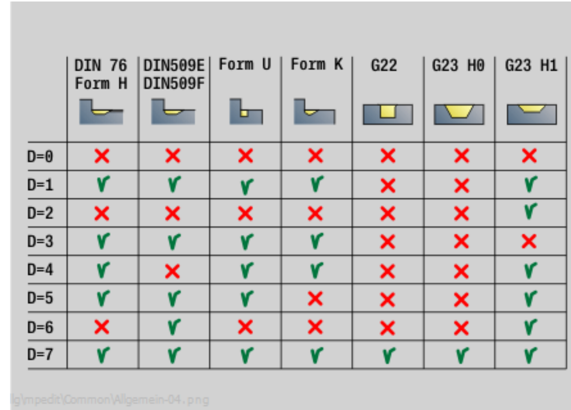
DI Axis-parallel oversize X

DK Axis-parallel oversize Z

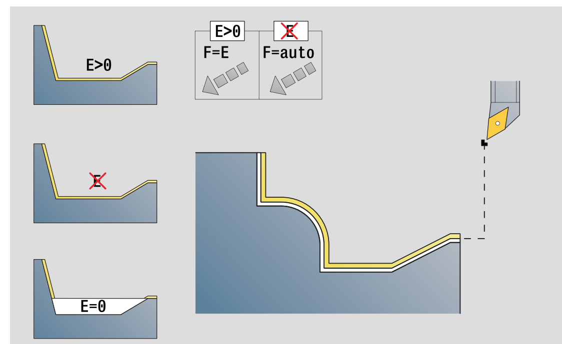
Further forms: see page 58

If feed rate reduction is active, at least four spindle revolutions are used to machine every "small" contour element.

With the address Dxx you activate an additive compensation for the entire cycle run. The additive compensation is switched off again at the end of the cycle. You edit additive compensation values in the Program Run mode of operation.



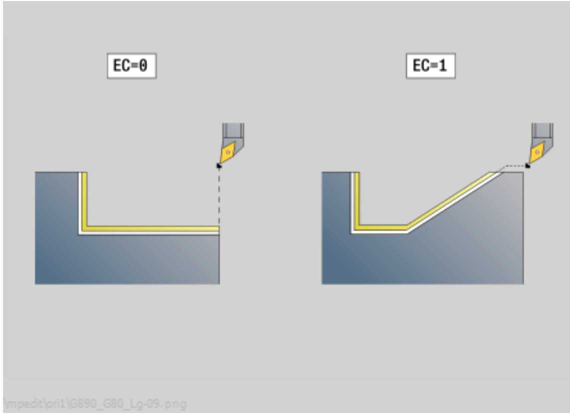
	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓

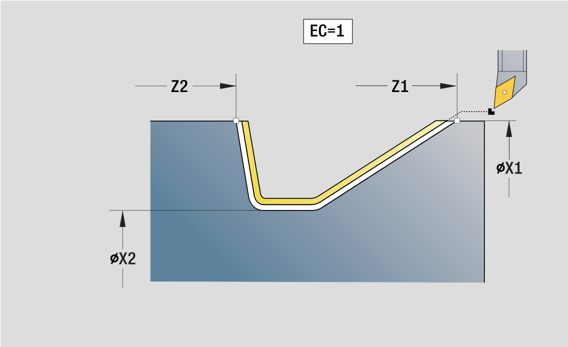


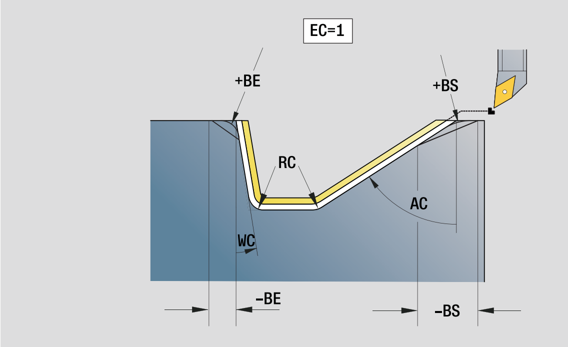
"Longitudinal finishing with direct contour input" unit

The unit finishes the contour defined by the parameters in one pass. In **EC** you define whether you want to machine a normal or a plunging contour.

Unit name: G890_G80_L / Cycle: G890 (see page 282)







Contour form

EC Type of contour

- 0: Normal contour
- 1: Plunging contour

X1, Z1 Contour starting point

X2, Z2 Contour end point

RC Rounding: Radius of contour corner

AC Start angle: Angle of the first contour element (range: 0° < AC < 90°)

WC End angle: Angle of the last contour element (range: 0° < WC < 90°)

BS Chamfer/radius at start

- BS>0: Radius of rounding arc
- BS<0: Section length of chamfer

BE Chamfer/radius at end

- BE>0: Radius of rounding arc
- BS<0: Section length of chamfer

Cycle form

E Plunging behavior

- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

B Switch on TRC (type of tool radius compensation)

- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

DXX Additive correction numbers 1 – 16

G58 Contour-parallel oversize (radius)

Further forms: see page 58

> With the address Dxx you activate an additive compensation for the entire cycle run. The additive compensation is switched off again at the end of the cycle. You edit additive compensation values in the Program Run mode of operation.

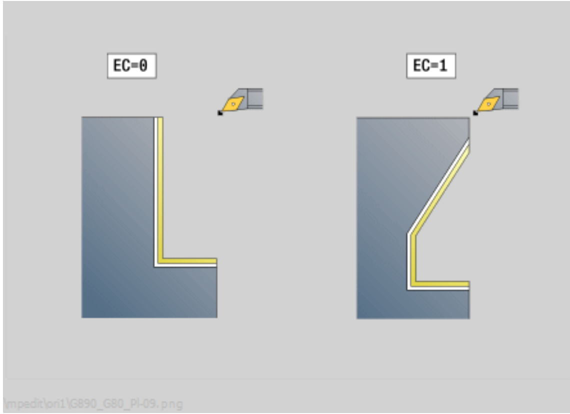
Access to the technology database:

- Machining operation: Finishing
- Affected parameters: F, S, E

"Transverse finishing with direct contour input" unit

The unit finishes the contour defined by the parameters in one pass. In **EC** you define whether you want to machine a normal or a plunging contour.

Unit name: G890_G80_P / Cycle: G890 (see page 282)



Contour form

EC — Type of contour
- 0: Normal contour
- 1: Plunging contour

X1, Z1 — Contour starting point

X2, Z2 — Contour end point

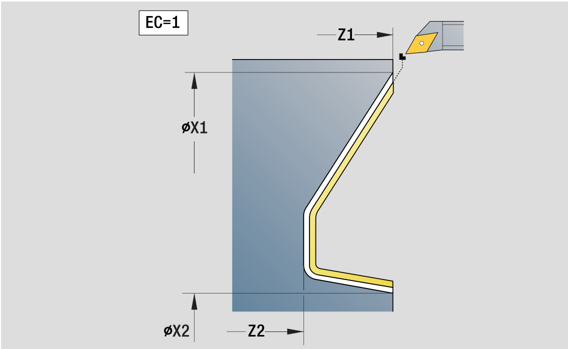
RC — Rounding: Radius of contour corner

AC — Start angle: Angle of the first contour element (range: 0° < AC < 90°)

WC — End angle: Angle of the last contour element (range: 0° < WC < 90°)

BS — Chamfer/radius at start:
- BS>0: Radius of rounding arc
- BS<0: Section length of chamfer

BE — Chamfer/radius at end
- BE>0: Radius of rounding arc
- BS<0: Section length of chamfer



Cycle form

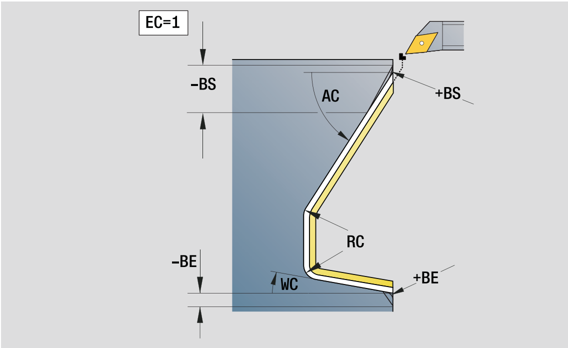
E — Plunging behavior
- E>0: Plunging feed rate for declining contour elements. Descending contour elements are machined.
- No input: The plunging feed rate is reduced during machining of declining contour elements by up to 50%. Descending contour elements are machined.

B — Switch on TRC (type of tool radius compensation)
- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

DXX — Additive correction numbers 1 – 16

G58 — Contour-parallel oversize (radius)

Further forms: see page 58



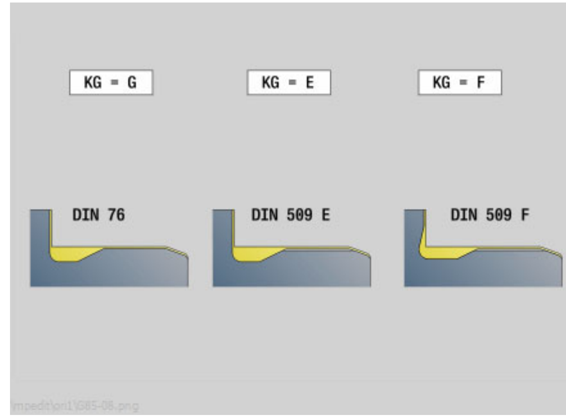
With the address Dxx you activate an additive compensation for the entire cycle run. The additive compensation is switched off again at the end of the cycle. You edit additive compensation values in the Program Run mode of operation.

Access to the technology database:
- Machining operation: Finishing
- Affected parameters: F, S, E

"Relief turns (undercut) type E, F, DIN76" unit

The unit machines the undercut defined by **KG**, and then the plane surface. The cylinder chamfer is executed when you enter at least one of the parameters **1st cut length** or **1st cut radius.**

Unit name: G85x_DIN_E_F_G / Cycle: G85 (see page 306)



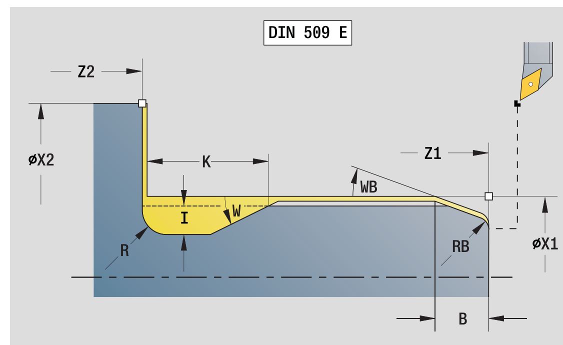
Overview form

KG — Type of relief turn (undercut)
- E: DIN 509 type E; Cycle G851 (see page 308)
- F: DIN 509 type F; Cycle G852 (see page 309)
- G: DIN 76 type G (thread undercut); Cycle G853 (see page 310)

X1, Z1 — Contour starting point (X1: diameter value)
X2, Z2 — Contour end point (X2: diameter value)
App — Approach see page 63

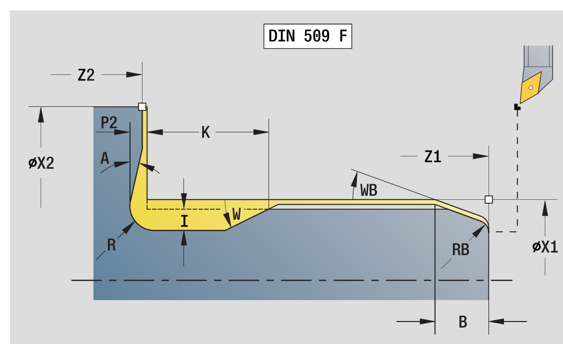
Parameters on the "Type E" form

I — Undercut depth (default: value from standard table)
K — Undercut length (default: value from standard table)
W — Undercut angle (default: standard table 15°)
R — Undercut radius (default: value from standard table)
H — Type of departure
- 0: To the starting point
- 1: Plane surface end



Parameters on the "Type F" form

I — Undercut depth (default: value from standard table)
K — Undercut length (default: value from standard table)
W — Undercut angle (default: standard table 15°)
R — Undercut radius (default: value from standard table)
P2 — Face depth (default: value from standard table)
A — Face angle (default: 8° from standard table)
H — Type of departure
- 0: To the starting point
- 1: Plane surface end



Access to the technology database:
- Machining operation: Finishing
- Affected parameters: F, S, E

Parameters on the "Type G" form

FP Thread pitch
I Undercut diameter (default: value from standard table)
K Undercut length (default: value from standard table)
W Undercut angle (default: value from standard table 30°)
R Undercut radius (default: value from standard table)
P1 Undercut oversize

- No input: Machining in one cut
- P1>0: Division into pre-turning and finish-turning; P1 is the longitudinal oversize; the transverse oversize is always 0.1 mm

H Type of departure

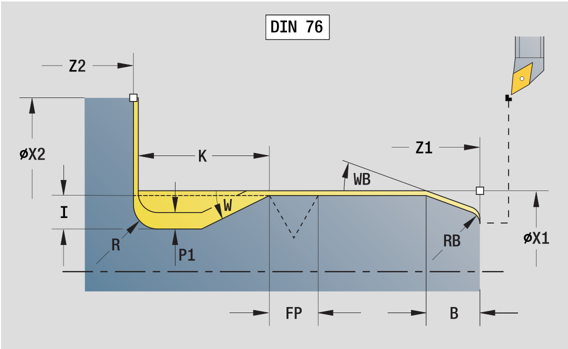
- 0: To the starting point
- 1: Plane surface end

Additional parameters for "cylinder first cut"

B Cylinder 1st cut length (no input: no cylinder start chamfer)
WB 1st cut angle (default: 45°)
RB Positive value: First cut radius, negative value: chamfer (no input: no element)
E Reduced feed rate for plunging and the first cut (default: active feed rate)
U Grinding oversize for cylinder

Further forms: see page 58

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- Parameters that are not programmed are automatically calculated by the Control from the standard table.



Access to the technology database:

- Machining operation: Finishing
- Affected parameters: F, S, E

"Measuring cut" unit

The unit performs a cylindrical measuring cut with the length defined in the cycle, moves to the breakpoint for measuring and stops the program. After the program was stopped, you can manually measure the workpiece.

Unit name: MEASURE_G809 / Cycle: G809 (see page 285)

Contour form

EC Machining location

- 0: Outside
- 1: Inside

XA, ZA Contour starting point

R Measuring cut length

P Measuring cut oversize

O Approach angle: If an approach angle is entered, the cycle positions the tool over the starting point taking into account the safety clearance and from there plunges at the specified angle to the diameter to be measured.

ZR Workpiece blank starting point: Collision-free approach for inside machining

Cycle form

QC Machining direction

- 0: -Z
- 1: +Z

V Measuring cut counter: Number of workpieces after which a measurement is performed

D Additive correction numbers 1 – 16

WE Directions

- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X

Xi, Zi: Additive correction numbers 1 – 16

AX Departing position X

Further forms: see page 58

2.8 Units—Threads

Overview of thread units

- **"Thread, direct"** cuts a simple internal or external thread in longitudinal direction.
- **"ICP thread"** cuts a single or multi-start internal or external thread in longitudinal or transverse direction. The contour on which the thread is cut is defined with ICP.
- **"API thread"** cuts a single or multi-start API thread. The depth of thread decreases at the overrun at the end of thread.
- **"Tapered thread"** cuts a single or multi-start tapered internal or external thread.

Handwheel superimposition

If your machine features handwheel superimposition, you can overlap axis movements during thread cutting in a limited area:

- **X direction**: Maximum programmed thread depth depending on the current cutting depth
- **Z direction**: +/- a fourth of the thread pitch

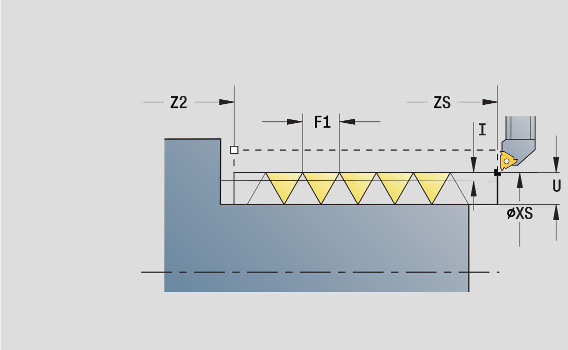
Machine and control must be specially prepared by the machine tool builder for use of this cycle. Refer to your machine manual.

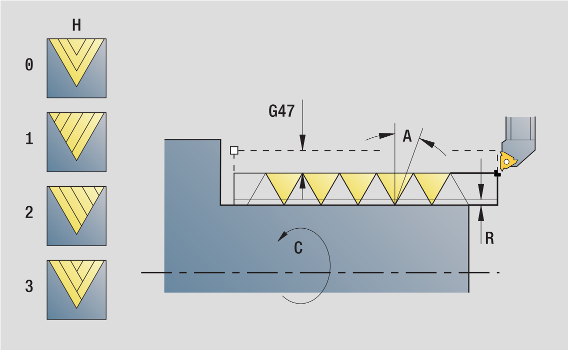
Remember that position changes resulting from handwheel superimposition are no longer effective after the cycle end or the "last cut" function.

"Thread, direct" unit

The unit cuts a simple internal or external thread in longitudinal direction.

Unit name: G32_MAN / Cycle: G32 (see page 297)





Thread form

O	Thread location ■ 0: Internal thread (infeed in +X) ■ 1: External thread (infeed in –X)
APP	Approach see page 63
XS	Start diameter
ZS	Starting position Z
Z2	End point of thread
F1	Thread pitch
U	Thread depth (automatically for metric ISO threads)
I	Maximum infeed (radius)
IC	Number of cuts (only if I is not programmed and the infeed V is 0 or 1)
KE	Run-out position ■ 0: At the end of the threading cut ■ 1: At the start of the threading cut
K	Run-out length

Access to the technology database:

- Machining operation: Thread cutting
- Affected parameters: F, S

Cycle form

H Kind of displacement (type of offset; offset between the individual infeeds in cutting direction)

- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

V Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110

A Approach angle (angle of infeed; reference in X axis 0°<A<60°, default 30°)

R Remaining cut depth (only with V=4)

C Starting angle

D No. of gears (threads per unit)

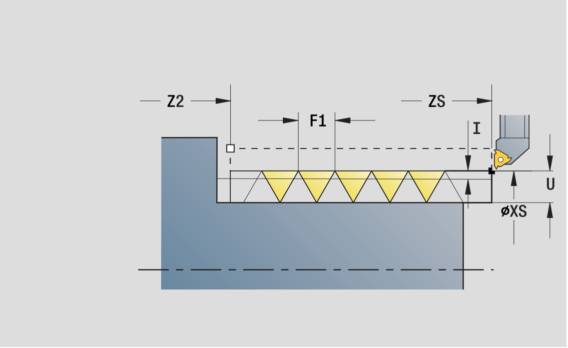
Q No. no load (number of dry runs)

Further forms: see page 58

"ICP thread" unit

The unit cuts a single or multi-start internal or external thread in longitudinal or transverse direction. The contour on which the thread is cut is defined with ICP.

Unit name: G31_ICP / Cycle: G31 (see page 293)



Thread form

FK Auxiliary contour: see page 60

NS Starting block no. of contour

NE End block no. of contour

O1 Machine form element

- 0: No machining
- 1: At beginning
- 2: At end
- 3: At beginning and end
- 4: Only chamfer and rounding arc

O Thread location

- 0: Internal thread (infeed in +X)
- 1: External thread (infeed in –X)

Access to the technology database:

- Machining operation: Thread cutting
- Affected parameters: F, S

J1 Thread orientation

- From 1st contour element
- 0: Longitudinal
- 1: Transverse

F1 Thread pitch

U Thread depth (automatically for metric ISO threads)

A Approach angle (angle of infeed; reference in X axis 0°<A>60°, default 30°)

D No. of gears (threads per unit)

K Run-out length

Cycle form

H Kind of displacement (type of offset; offset between the individual infeeds in cutting direction)

- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

V Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110

R Remaining cut depth (only with V=4)

I Maximum infeed (radius)

IC Number of cuts (only if I is not programmed)

B Run-in length

P Overrun length

C Starting angle

Q No. no load (number of dry runs)

Further forms: see page 58

"API thread" unit

This unit cuts a single or multi-start API thread. The depth of thread decreases at the overrun at the end of thread.

Unit name: G352_API / Cycle: G352 (see page 302)

Thread form

O Thread location

- 0: Internal thread (infeed in +X)
- 1: External thread (infeed in –X)

X1, Z1 Starting point of thread (X1: diameter value)

X2, Z2 End point of thread (X2: diameter value)

W Taper angle (reference: Z axis: –45°<W<45°)

WE Run-out angle (reference: Z axis: 0°<WE<90°, default: 12°)

F1 Thread pitch

U Thread depth (automatically for metric ISO threads)

Cycle form

I Maximum infeed (radius)

H Kind of displacement (type of offset; offset between the individual infeeds in cutting direction)

- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

V Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110

A Approach angle (angle of infeed; reference in X axis 0°>A>60°, default 30°)

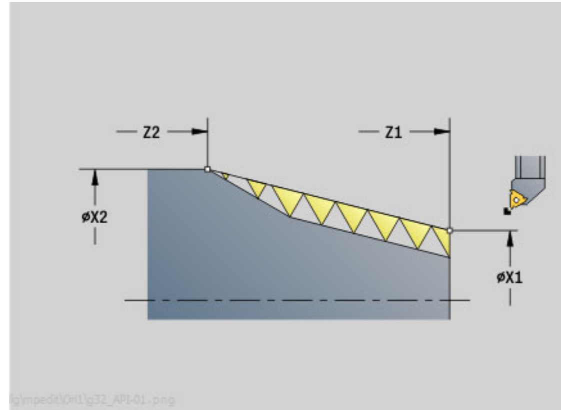
R Remaining cut depth (only with V=4)

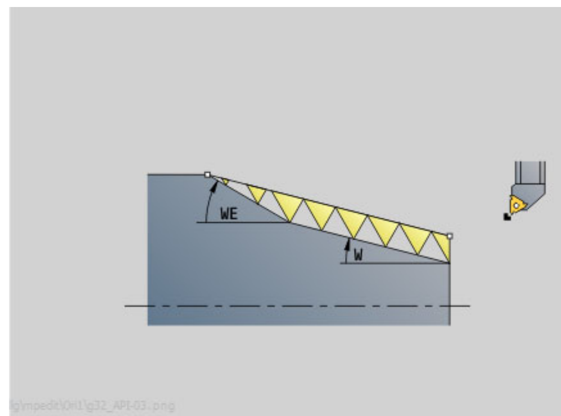
C Starting angle

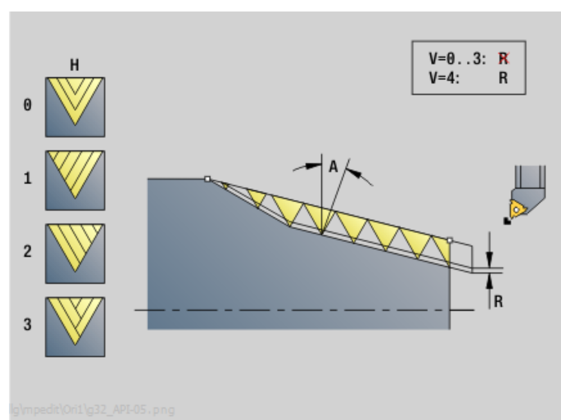
D No. of gears (threads per unit)

Q No. no load (number of dry runs)

Further forms: see page 58







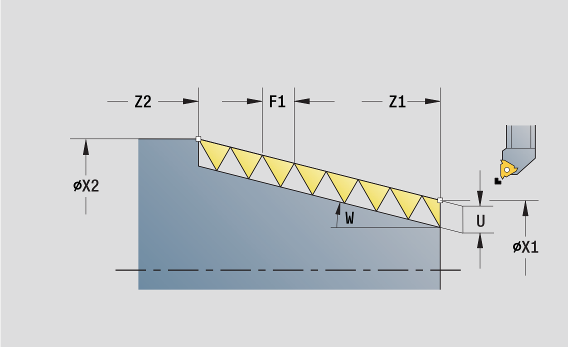
Access to the technology database:

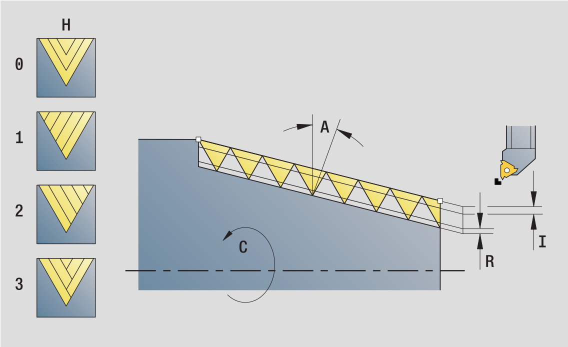
- Machining operation: Thread cutting
- Affected parameters: F, S

"Tapered thread" unit

The unit cuts a single or multi-start tapered internal or external thread.

Unit name: G32_KEG / Cycle: G32 (see page 297)





Thread form

O Thread location

- 0: Internal thread (infeed in +X)
- 1: External thread (infeed in –X)

X1, Z1 Starting point of thread (X1: diameter value)

X2, Z2 End point of thread (X2: diameter value)

W Taper angle (reference: Z axis: –45°<W<45°)

F1 Thread pitch

U Thread depth (automatically for metric ISO threads)

KE Run-out position

- 0: At the end of the threading cut
- 1: At the start of the threading cut

K Run-out length

Access to the technology database:

- Machining operation: Thread cutting
- Affected parameters: F, S

Cycle form

I	Maximum infeed (radius)
IC	Number of cuts (only if I is not programmed)
H	Kind of displacement (type of offset; offset between the individual infeeds in cutting direction)

- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

V Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110

A	Approach angle (angle of infeed; reference in X axis 0°<A<60°, default 30°)
R	Remaining cut depth (only with V=4)
C	Starting angle
D	No. of gears (threads per unit)
Q	No. no load (number of dry runs)

Further forms: see page 58

2.9 Units—Milling, face

"Slot, face" unit

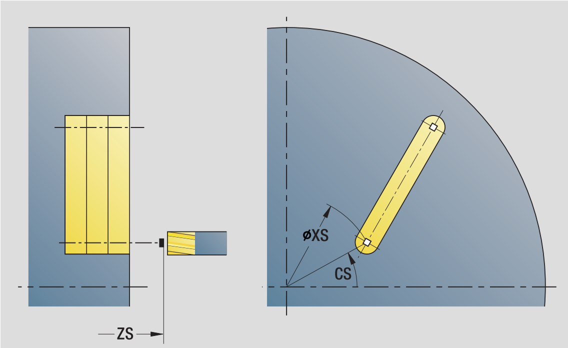
The unit mills a slot from the starting position to the end point on the face of the workpiece. The slot width equals the diameter of the milling cutter.

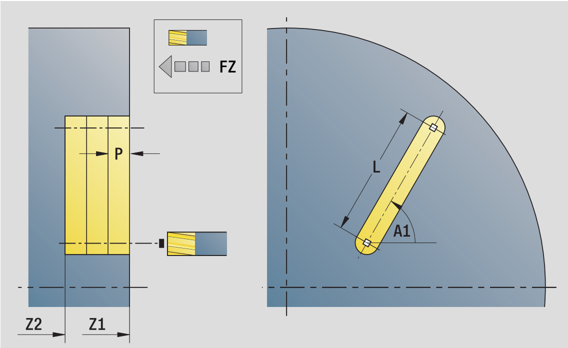
Unit name: G791_Nut_Stirn_C / Cycle: G791 (see page 339)

Cycle form

Z1	Milling top edge
Z2	Milling floor
L	Slot length
A1	Angle to X axis
X1, C1	Polar slot target point
XK, YK	Cartesian slot target point
P	Maximum infeed
FZ	Infeed rate

Further forms: see page 58





Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Linear slot pattern, face" unit

The unit machines a linear slot pattern in which the individual features are arranged at a regular spacing on the face of the workpiece. The starting points of the slots correspond to the pattern positions. You define the length and the position of the slots in the unit. The slot width equals the diameter of the milling cutter.

Unit name: G791_Lin_Stirn_C / Cycle: G791 (see page 339)

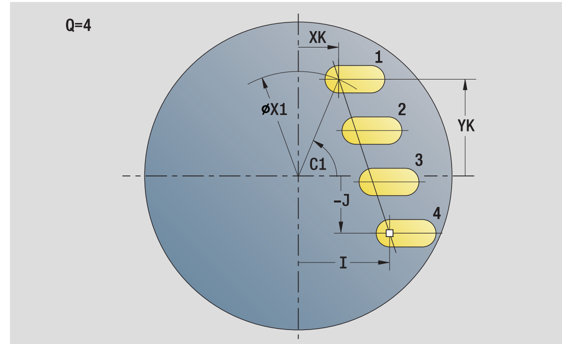
Pattern form

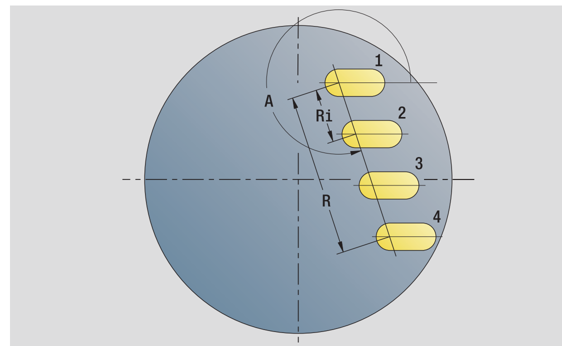
Q	Number of slots
X1, C1	Polar starting point
XK, YK	Cartesian starting point
I, J	End point (XK, YK)
Ii, Ji:	Distance (XKi, YKi)
R	Distance to first/last contour
Ri	Incremental distance
A	Pattern angle (reference is XK axis)

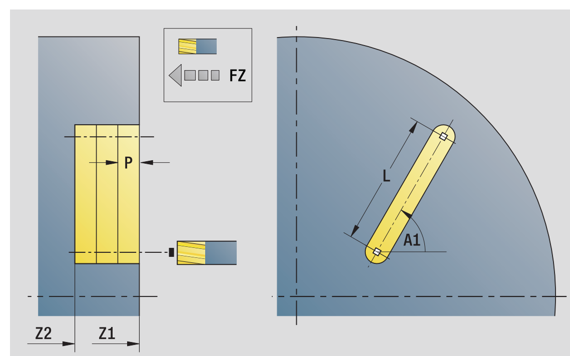
Cycle form

Z1	Milling top edge
Z2	Milling floor
L	Slot length
A1	Angle to X axis
P	Maximum infeed
FZ	Infeed rate

Further forms: see page 58







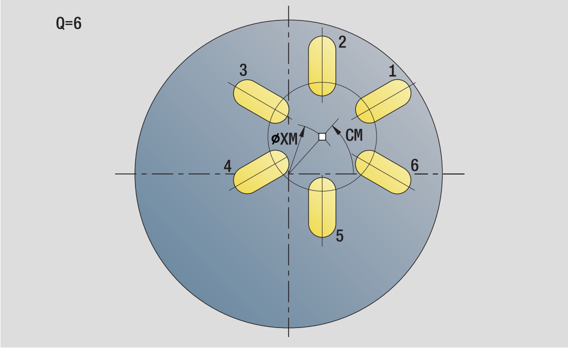
Access to the technology database:

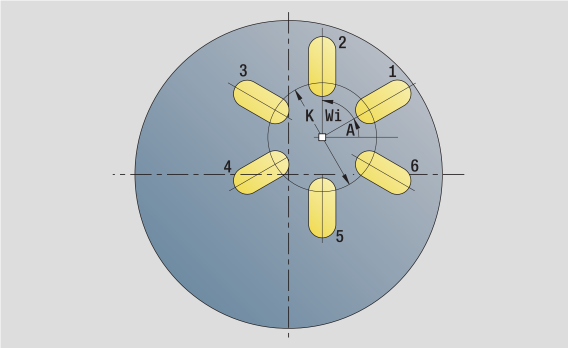
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

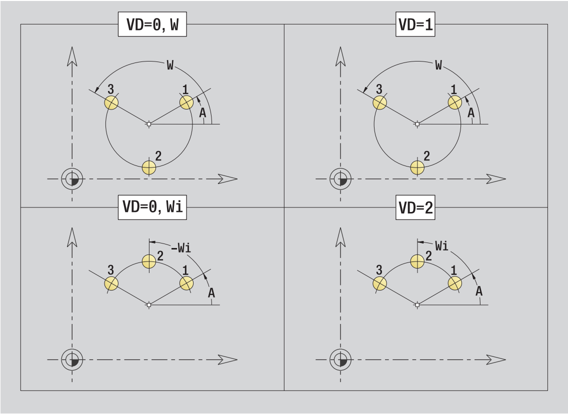
"Circular slot pattern, face" unit

The unit machines a circular slot pattern in which the individual features are arranged at a regular spacing on the face of the workpiece. The starting points of the slots correspond to the pattern positions. You define the length and the position of the slots in the unit. The slot width equals the diameter of the milling cutter.

Unit name: G791_Cir_Stirn_C / Cycle: G791 (see page 339)







Pattern form

Q	Number of slots
XM, CM	Polar center point
XK, YK	Cartesian center point
A	Starting angle
Wi	Angle increment
K	Pattern diameter
W	End angle
V	Rotation direction (default: 0)

- VD=0, without W: Figures are arranged on a full circle
- VD=0, with W: Figures are arranged on the longer circular arc
- VD=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- VD=1, with W: Clockwise
- VD=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- VD=2, with W: Counterclockwise
- VD=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)

Cycle form

Z1	Milling top edge
Z2	Milling floor
L	Slot length
A1	Angle to X axis
P	Maximum infeed
FZ	Infeed rate

Further forms: see page 58

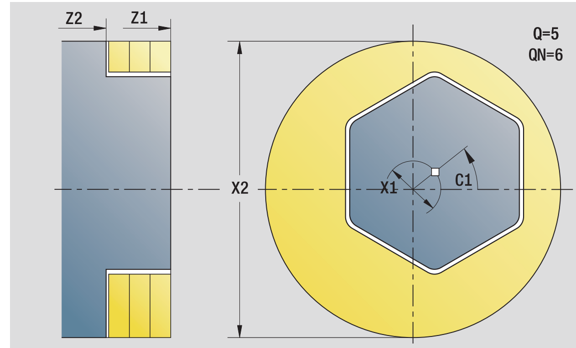
Access to the technology database:

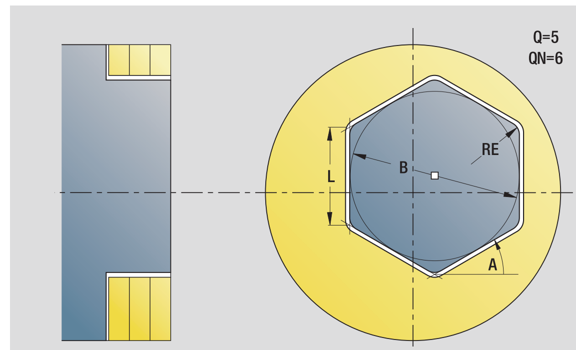
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Face milling" unit

Depending on **Q**, the unit mills surfaces or the defined figure. The unit cuts the material around the figures.

Unit name: G797_Stirnfr_C / Cycle: G797 (see page 345)





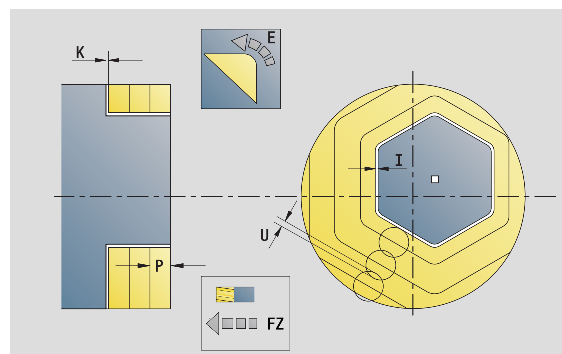


Figure form

Q Type of figure

- 0: Full circle
- 1: Single surface
- 2: Width across flats
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

QN Number of polygon corners (only with Q=5 polygon)
X1 Diameter of figure center
C1 Angle of figure center
Z1 Milling top edge
Z2 Milling floor
X2 Limit diameter
L Length of edge
B Width/Width across flats
RE Rounding radius
A Angle to X axis

Cycle form

QK Machining operation

- Roughing
- Finishing

J Milling direction

- 0: Unidirectional
- 1: Bidirectional

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed
I Contour-parallel oversize
K Infeed-direction oversize
FZ Infeed rate
E Reduced feed rate
U Overlap factor

Further forms: see page 58

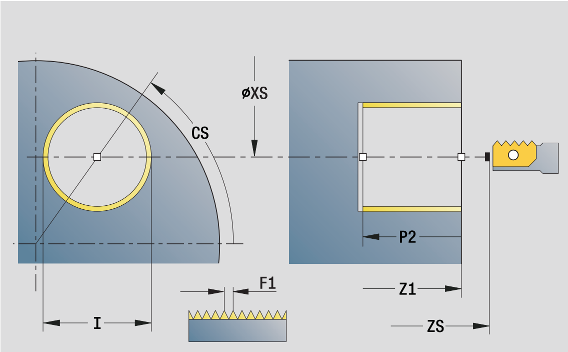
Access to the technology database:

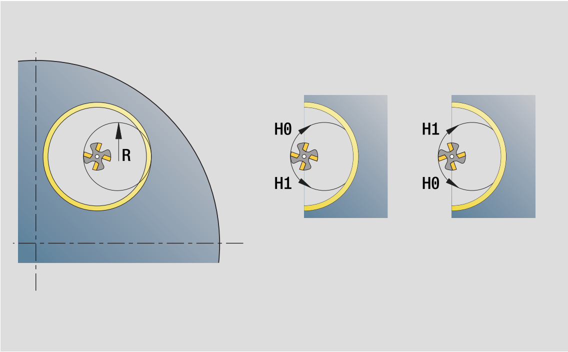
- Machining operation: Finish-milling
- Affected parameters: F, S, FZ, P

"Thread milling" unit

The unit mills a thread in existing holes.

Place the tool on the center of the hole before calling G799. The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one rotation or, with single-point tools, in several rotations.





Unit name: G799_Gewindefr_C / Cycle: G799 (see page 328)

Position form

Z1 Start point drill (starting point of hole)
P2 Thread depth
I Thread diameter
F1 Thread pitch

Cycle form

J Direction of thread
- 0: Right-hand thread
- 1: Left-hand thread

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

V Milling method
- 0: The thread is milled in a 360-degree helix
- 1: The thread is milled in several helical paths (single-point tool)

R Approach radius

Further forms: see page 58

Access to the technology database:

- Machining operation: Finish-milling
- Affected parameters: F, S

"Contour milling, figures, face" unit

The unit mills the contour defined by **Q** on the face of the workpiece.

Unit name: G840_Fig_Stirn_C/ Cycle: G840 (see page 351)

Figure form

Q	Type of figure

- 0: Full circle
- 1: Linear slot
- 2: Circular slot
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

QN	Number of polygon corners—only with Q=5 (polygon)
X1	Diameter of figure center
C1	Angle of figure center
Z1	Milling top edge
P2	Depth of figure
L	Edge length / width across flats

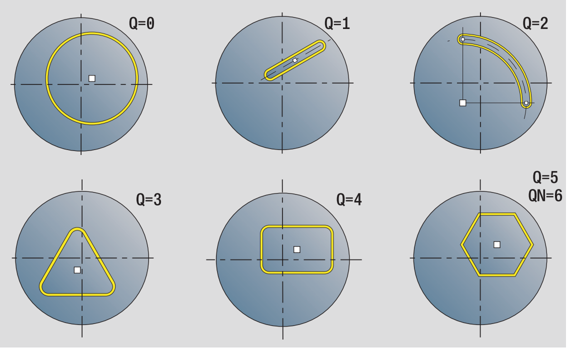
- L>0: Edge length
- L<0: Width across flats (inside diameter) for polygon

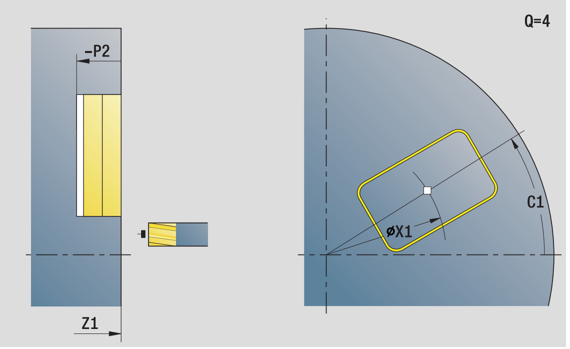
B	Rectangle width
RE	Rounding radius
A	Angle to X axis
Q2	Rotational direction of slot—only if Q=2 (circular slot)

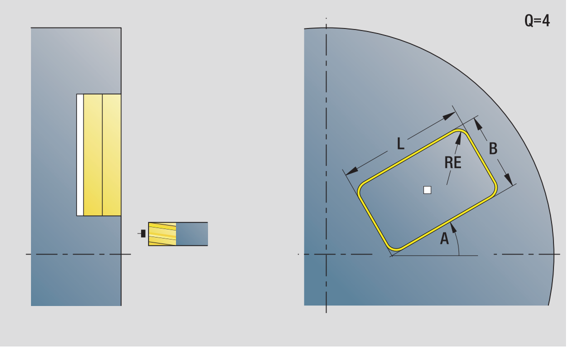
- cw: In clockwise direction
- ccw: In counterclockwise direction

W	Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.







Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

Cycle form

JK Cutter position

- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed

I Contour-parallel oversize

K Infeed-direction oversize

FZ Infeed rate

E Reduced feed rate

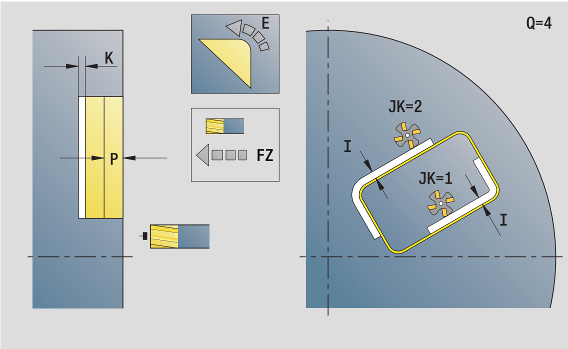
R Approach radius

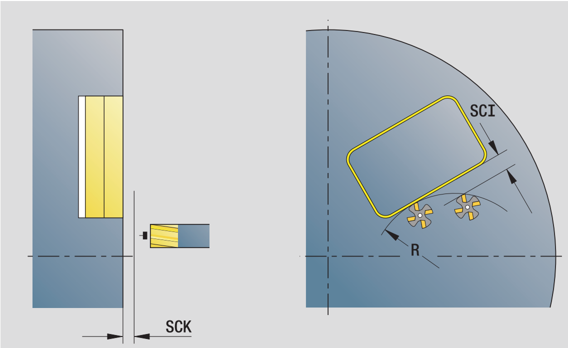
O Plunging behavior

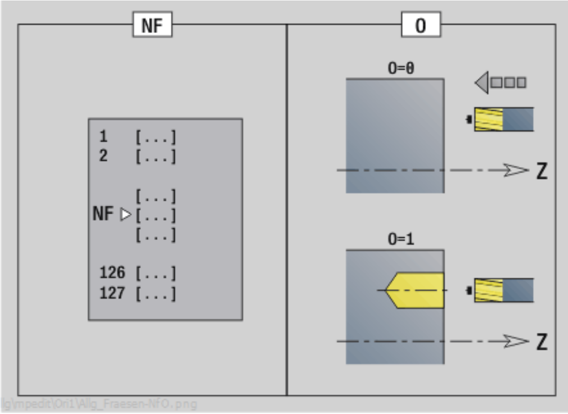
- 0: Straight (vertical plunge)—The cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour.
- 1: In predrilling—The cycle positions the tool above the hole; the tool plunges and mills the contour.

NF Position mark (only if O=1)

Further forms: see page 58



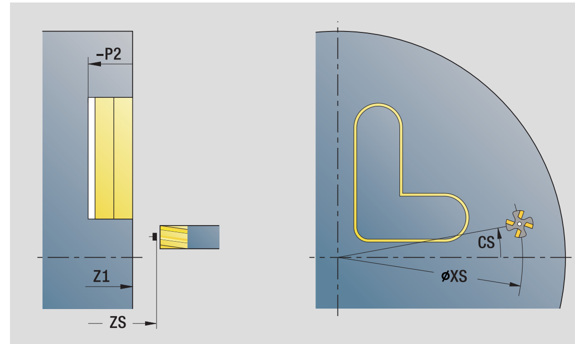


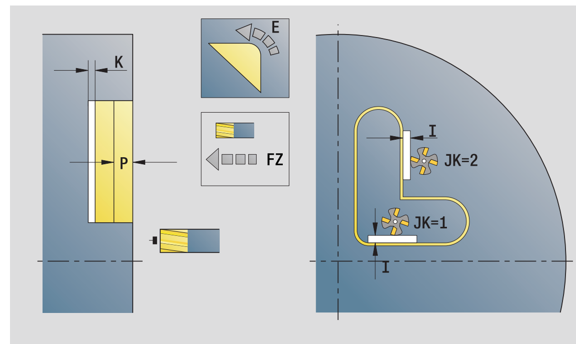


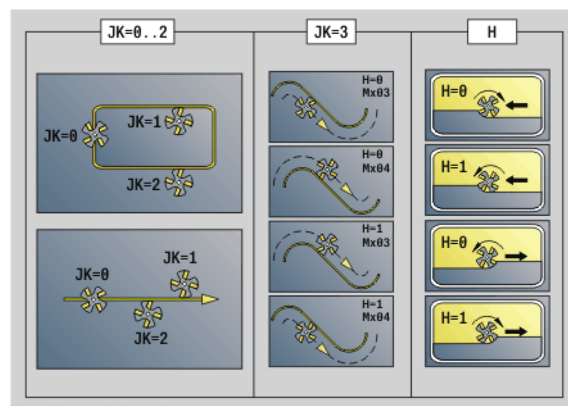
"ICP contour milling, face" unit

The unit mills the contour defined with ICP on the face of the workpiece.

Unit name: G840_Kon_C_Stirn / Cycle: G840 (see page 351)







Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
Z1	Milling top edge
P2	Depth of contour

Cycle form

JK Cutter position

- 0: On the contour
- 1, closed contour: Within the contour
- 1, open contour: Left of the contour
- 2, closed contour: Outside the contour
- 2, open contour: Right of the contour
- 3: Depending on H and MD

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

P	Maximum infeed
I	Contour-parallel oversize
K	Infeed-direction oversize
FZ	Infeed rate
E	Reduced feed rate
R	Approach radius
O	Plunging behavior

- 0: Straight (vertical plunge)—The cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour.
- 1: In predrilling—The cycle positions the tool above the hole; the tool plunges and mills the contour.

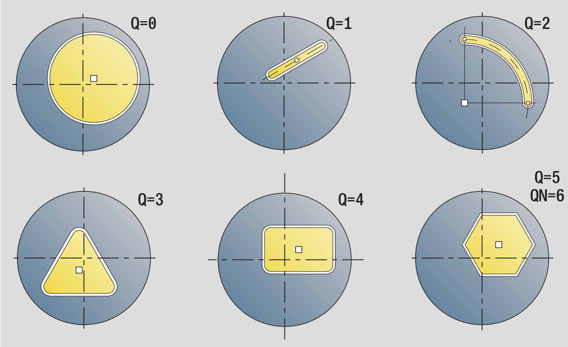
NF	Position mark (only if O=1)
RB	Return plane

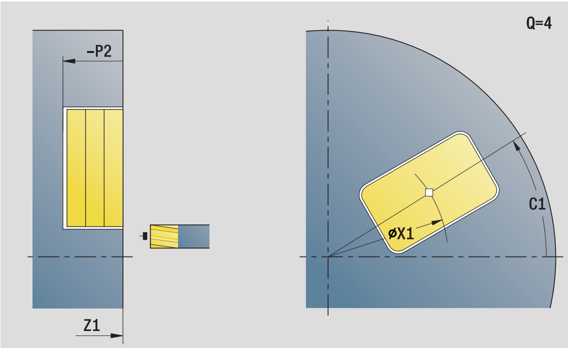
Further forms: see page 58

"Pocket milling, figures, face" unit

The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.

Unit name: G84x_Fig_Stirn_C / Cycles: G845 (see page 360); G846 (see page 364)





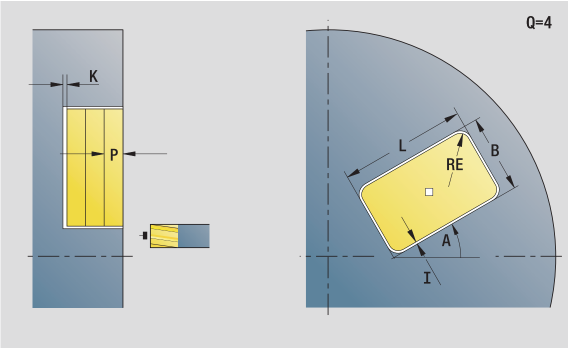


Figure form

Q Type of figure

- 0: Full circle
- 1: Linear slot
- 2: Circular slot
- 3: Triangle
- 4: Rectangle, square
- 5: Polygon

QN Number of polygon corners—only with Q=5 (polygon)
X1 Diameter of figure center
C1 Angle of figure center
Z1 Milling top edge
P2 Depth of figure
L Edge length / width across flats

- L>0: Edge length
- L<0: Width across flats (inside diameter) for polygon

B Rectangle width
RE Rounding radius
A Angle to X axis
Q2 Rotational direction of slot—only if Q=2 (circular slot)

- cw: In clockwise direction
- ccw: In counterclockwise direction

W Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.

Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

Cycle form

QK Machining operation and plunging strategy

- 0: Roughing
- 1: Finishing
- 2: Helical roughing, manual
- 3: Helical roughing, automatic
- 4: Reciprocating linear roughing, manual
- 5: Reciprocating linear roughing, automatic
- 6: Reciprocating circular roughing, manual
- 7: Reciprocating circular roughing, automatic
- 8: Plunge roughing at predrilling position
- 9: Finishing with 3-D approach arc

JT Machining direction

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed

I Contour-parallel oversize

K Infeed-direction oversize

FZ Infeed rate

E Reduced feed rate

R Approach radius

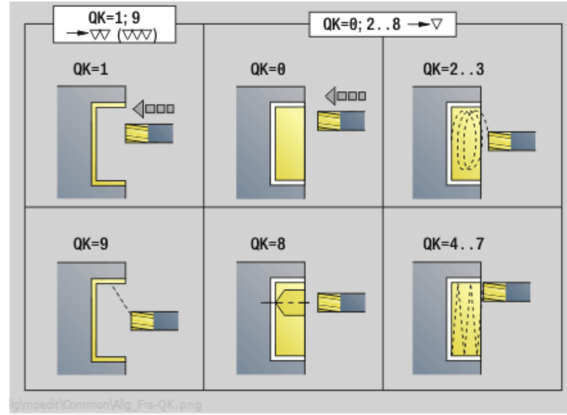
WB Plunging length

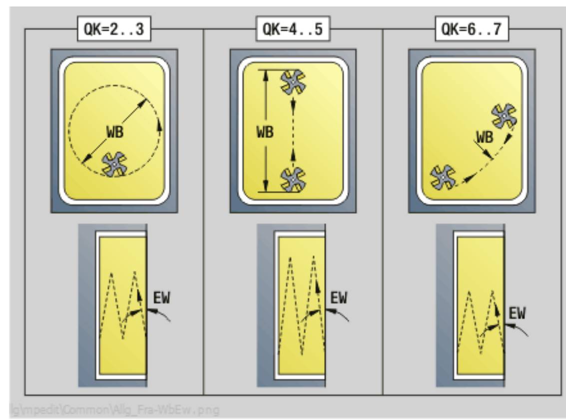
EW Plunging angle

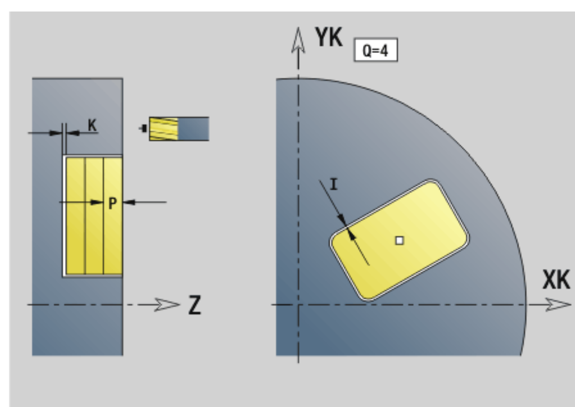
NF Position mark (only if QK=8)

U Overlap factor (default: 0.5)

Further forms: see page 58



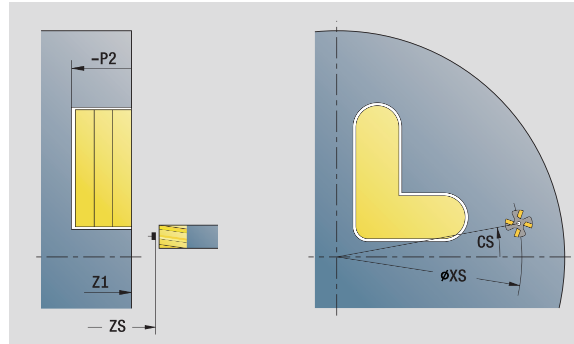


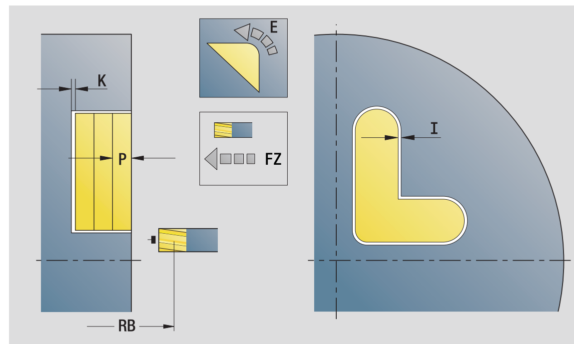


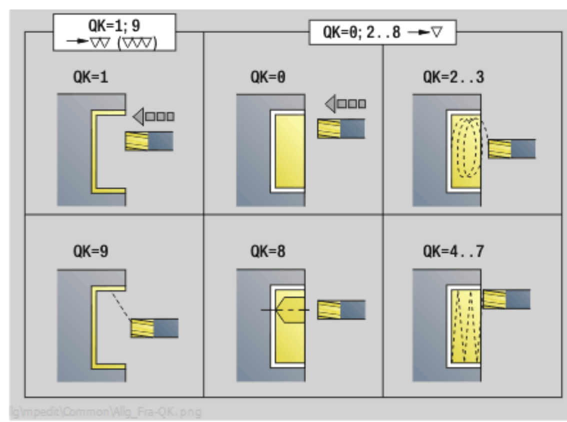
"ICP pocket milling, face" unit

The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.

Unit name: G845_Tas_C_Stirn / Cycles: G845 (see page 360); G846 (see page 364)







Contour form

FK see page 60
NS Starting block no. of contour
NE End block no. of contour
Z1 Milling top edge
P2 Depth of contour
NF Position mark (only if QK=8)

Cycle form

QK Machining operation and plunging strategy
- 0: Roughing
- 1: Finishing
- 2: Helical roughing, manual
- 3: Helical roughing, automatic
- 4: Reciprocating linear roughing, manual
- 5: Reciprocating linear roughing, automatic
- 6: Reciprocating circular roughing, manual
- 7: Reciprocating circular roughing, automatic
- 8: Plunge roughing at predrilling position
- 9: Finishing with 3-D approach arc

JT Machining direction
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed
I Contour-parallel oversize
K Infeed-direction oversize
FZ Infeed rate
E Reduced feed rate
R Approach radius
WB Plunging length
EW Plunging angle
U Overlap factor (default: 0.5)
RB Return plane

Further forms: see page 58

Access to the technology database:

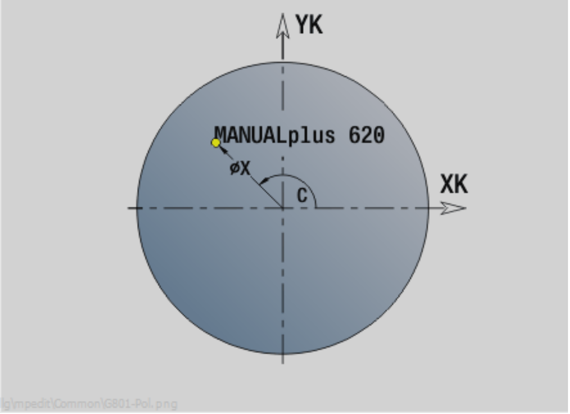
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

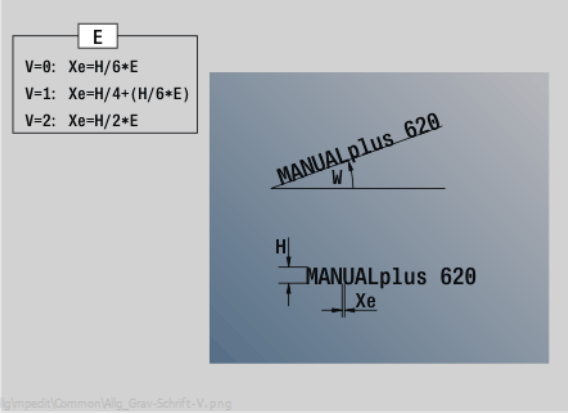
"Engraving, face" unit

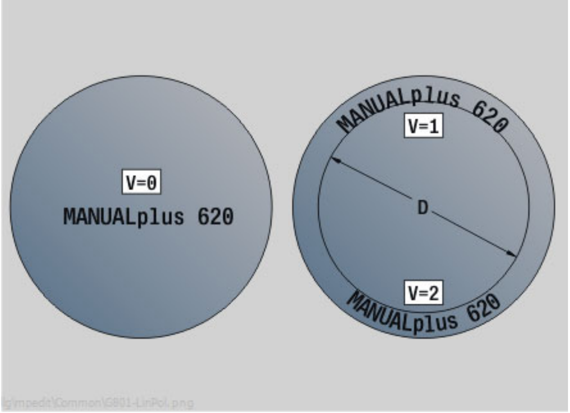
The unit engraves character strings in linear or polar layout on the face of the workpiece. Diacritics and special characters that you cannot enter in the smart.Turn editor can be defined, character by character, in **NF**. If you program "Continue from last text" (Q=1), tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.

Unit name: G801_GRA_STIRN_C / Cycle: G801 (see page 368)

Character set: see page 366







Position form

X, C	Polar starting point
XK, YK	Cartesian starting point
Z	End point. Z position, infeed depth during milling.
RB	Return plane

Cycle form

TXT	Text to be engraved
NF	Character number (character to be engraved)
H	Font height
E	Distance factor (for calculation see figure)
W	Inclination angle
FZ	Plunging feed rate factor (plunging feed rate = current feed rate * FZ)
V	Execution

- 0: Linear
- 1: Arched above
- 2: Arched below

D	Reference diameter
Q	Continue from last text

- 0 (No): Engraving starts at the starting point
- 1 (Yes): Engraving starts at the tool position

Further forms: see page 58

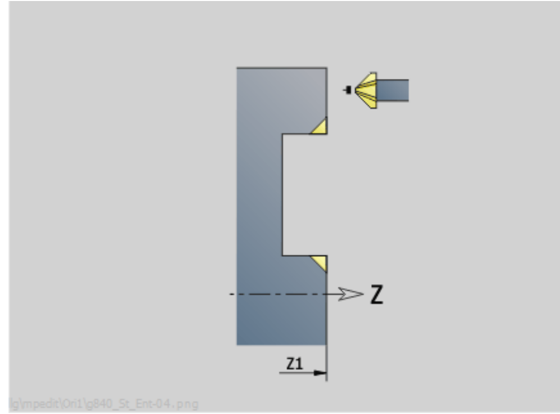
Access to the technology database:

- Machining operation: Engraving
- Affected parameters: F, S

"Deburring, face" unit

The unit deburrs the contour defined with ICP on the face of the workpiece.

Unit name: G840_ENT_C_STIRN / Cycle: G840 (see page 355)



Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
Z1	Milling top edge

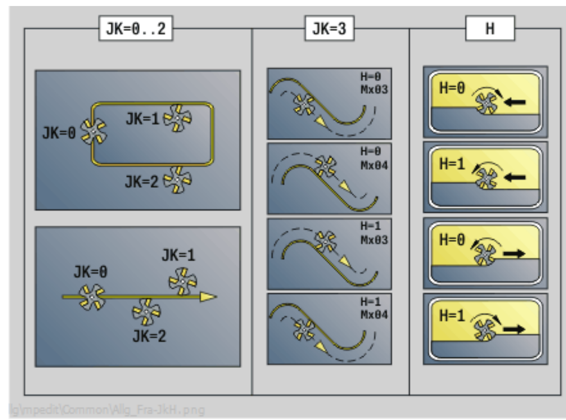
Cycle form

JK Cutter position

- JK=0: On the contour
- JK=1, closed contour: Within the contour
- JK=1, open contour: Left of the contour
- JK=2, closed contour: Outside the contour
- JK=2, open contour: Right of the contour
- JK=3: Depending on H and MD

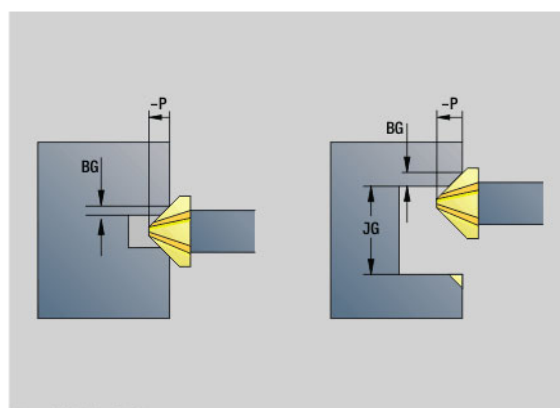
H Cutting direction

- 0: Up-cut milling
- 1: Climb milling



BG	Chamfer width
JG	Preparation diameter
P	Plunging depth (indicated as a negative value)
I	Contour-parallel oversize
R	Approach radius
FZ	Infeed rate
E	Reduced feed rate
RB	Return plane

Further forms: see page 58



Access to the technology database:

- Machining operation: Deburring
- Affected parameters: F, S

2.10 Units—Milling, lateral surface

"Slot, lateral surface" unit

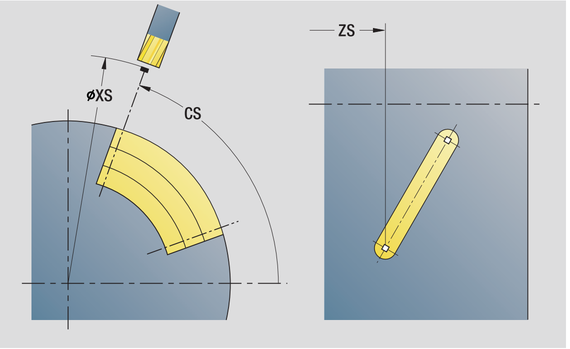
The unit mills a slot from the starting position to the end point on the lateral surface. The slot width equals the diameter of the milling cutter.

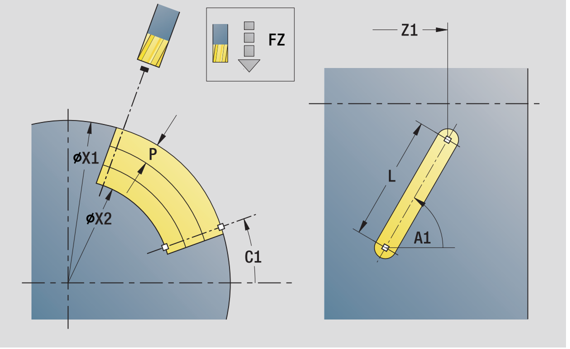
Unit name: G792_Nut_MANT_C / Cycle: G792 (see page 340)

Cycle form

X1	Milling top edge (diameter value)
X2	Milling floor (diameter)
L	Slot length
A1	Angle to Z axis
Z1, C1	Polar slot target point
P	Maximum infeed
FZ	Infeed rate

Further forms: see page 58





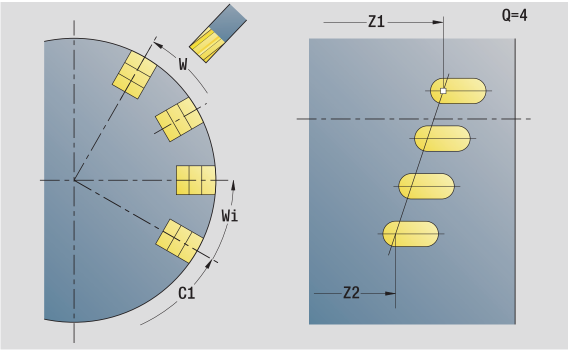
Access to the technology database:

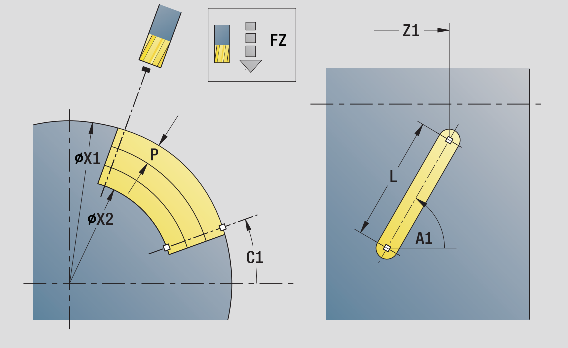
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Linear slot pattern, lateral surface" unit

The unit machines a linear slot pattern in which the individual features are arranged at a regular spacing on the lateral surface. The starting points of the slots correspond to the pattern positions. You define the length and the position of the slots in the unit. The slot width equals the diameter of the milling cutter.

Unit name: G792_Lin_Mant_C / Cycle: G792 (see page 340)





Pattern form

Q Number of slots
Z1, C1 Starting point of pattern
Wi Angle increment
W End angle
Z2 End point of pattern

Cycle form

X1 Milling top edge (diameter value)
X2 Milling floor (diameter)
L Slot length
A1 Angle to Z axis
P Maximum infeed
FZ Infeed rate

Further forms: see page 58

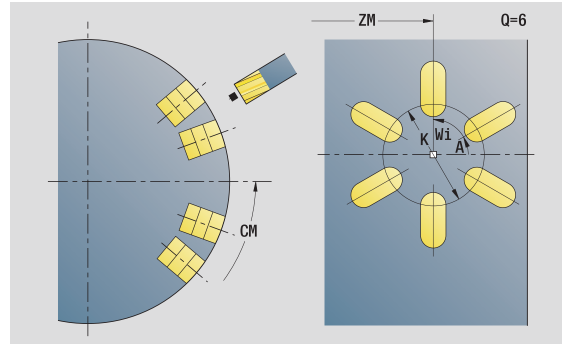
Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Circular slot pattern, lateral surface" unit

The unit machines a circular slot pattern in which the individual features are arranged at a regular spacing on the lateral surface. The starting points of the slots correspond to the pattern positions. You define the length and the position of the slots in the unit. The slot width equals the diameter of the milling cutter.

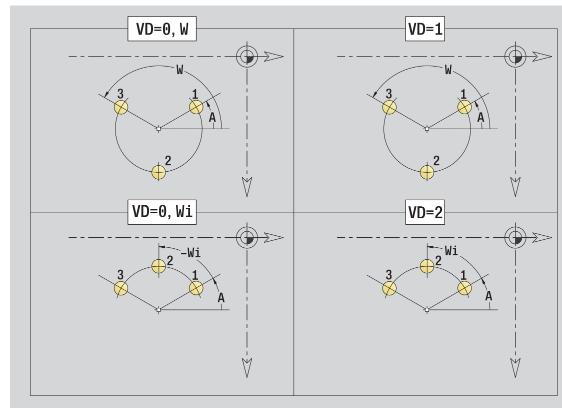
Unit name: G792_Cir_Mant_C / Cycle: G792 (see page 340)



Pattern form

Q	Number of slots
ZM, CM	Center point of pattern
A	Starting angle
Wi	Angle increment
K	Pattern diameter
W	End angle
V	Rotation direction (default: 0)

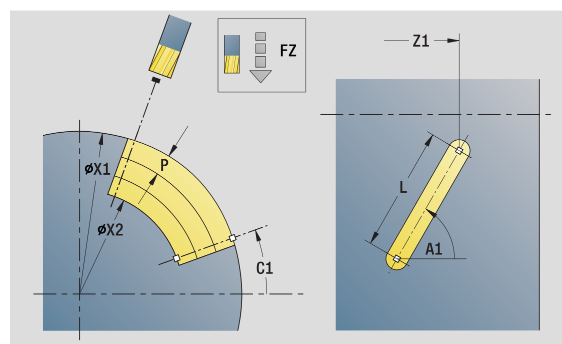
- VD=0, without W: Figures are arranged on a full circle
- VD=0, with W: Figures are arranged on the longer circular arc
- VD=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- VD=1, with W: Clockwise
- VD=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- VD=2, with W: Counterclockwise
- VD=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)



Cycle form

X1	Milling top edge (diameter value)
X2	Milling floor (diameter)
L	Slot length
A1	Angle to Z axis
P	Maximum infeed
FZ	Infeed rate

Further forms: see page 58



Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Helical slot milling" unit

The unit mills a helical slot. The slot width equals the diameter of the milling cutter.

Unit name: G798_Wendelnut_C / Cycle: G798 (see page 347)

Position form

X1 Thread diameter
C1 Starting angle
Z1 Starting point of thread
Z2 End point of thread
U Thread depth

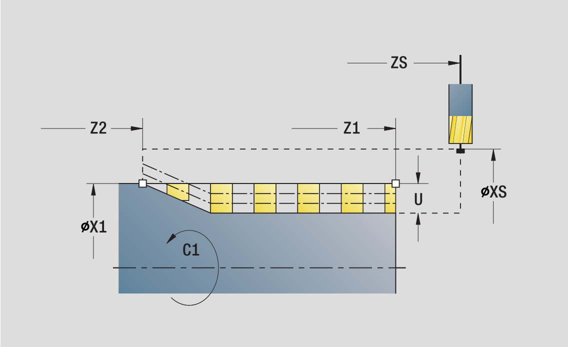
Cycle form

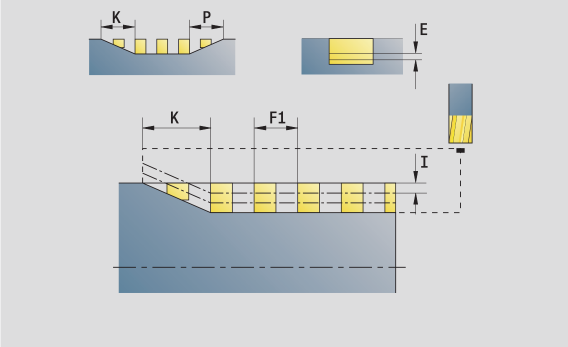
F1 Thread pitch
J Direction of thread:

- 0: Right-hand thread
- 1: Left-hand thread

D No. of gears (threads per unit)
P Run-in length
K Run-out length
I Maximum infeed
E Cutting depth reduction

Further forms: see page 58





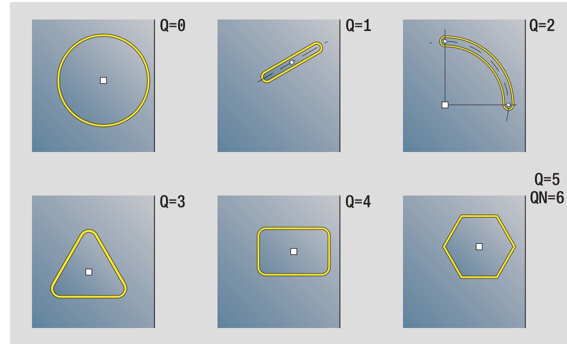
Access to the technology database:

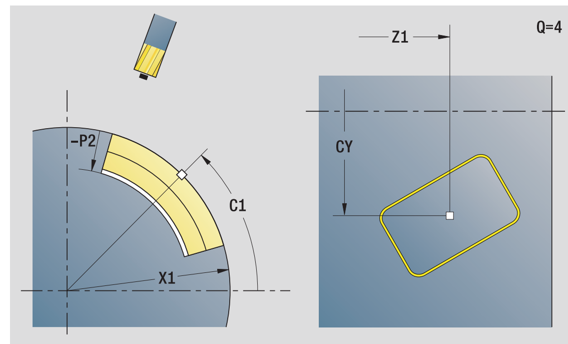
- Machining operation: Finish-milling
- Affected parameters: F, S

"Contour milling, figures, lateral surface" unit

The unit mills the contour defined by **Q** on the lateral surface.

Unit name: G840_Fig_Mant_C / Cycle: G840 (see page 351)





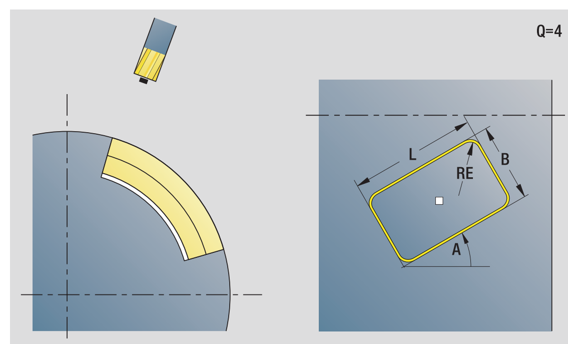


Figure form

Q	Type of figure ■ 0: Full circle ■ 1: Linear slot ■ 2: Circular slot ■ 3: Triangle ■ 4: Rectangle, square ■ 5: Polygon
QN	Number of polygon corners—only with Q=5 (polygon)
Z1	Figure center
C1	Angle of figure center
CY	Figure center of unrolled lateral surface
X1	Milling top edge
P2	Depth of figure
L	Edge length / width across flats ■ L>0: Edge length ■ L<0: Width across flats (inside diameter) for polygon
B	Rectangle width
RE	Rounding radius
A	Angle to Z axis
Q2	Rotational direction of slot—only if Q=2 (circular slot) ■ cw: In clockwise direction ■ ccw: In counterclockwise direction
W	Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.

Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

Cycle form

JK Cutter position

- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

H Cutting direction

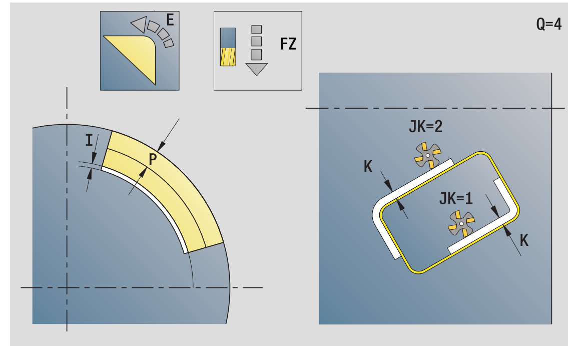
- 0: Up-cut milling
- 1: Climb milling

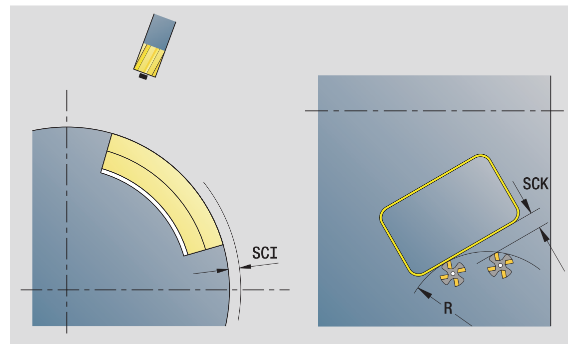
P Maximum infeed
I Infeed-direction oversize
K Contour-parallel oversize
FZ Infeed rate
E Reduced feed rate
R Approach radius
O Plunging behavior

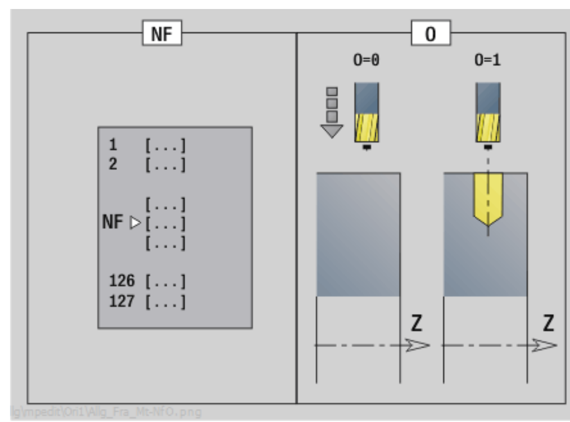
- 0: Straight (vertical plunge)—The cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour.
- 1: In predrilling—The cycle positions the tool above the hole; the tool plunges and mills the contour.

NF Position mark (only if O=1)

Further forms: see page 58



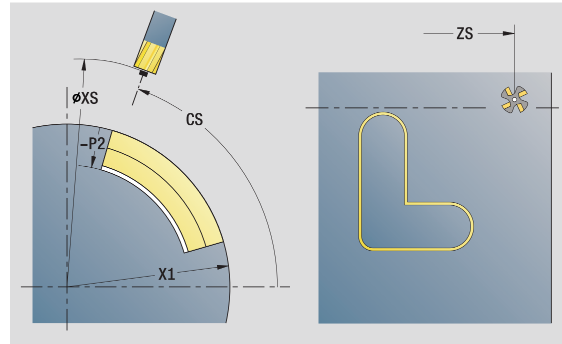


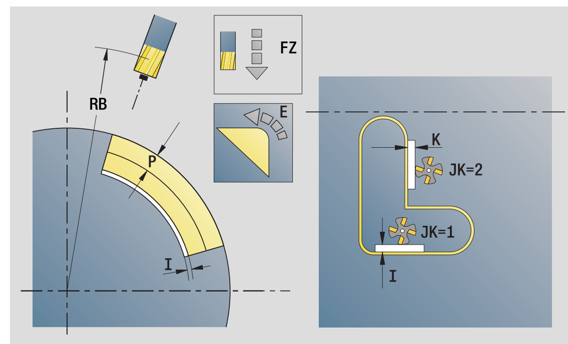


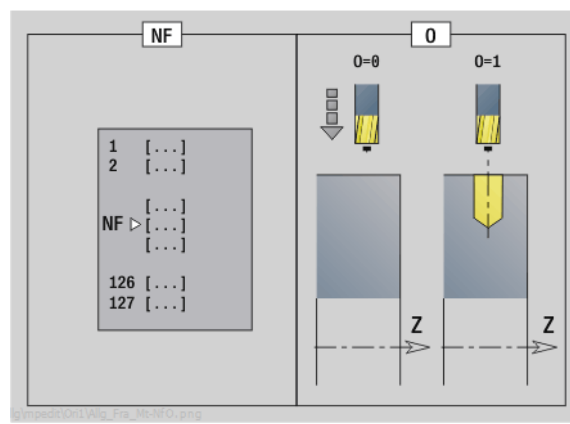
"ICP contour milling, lateral surface" unit

The unit mills the contour defined with ICP on the lateral surface.

Unit name: G840_Kon_C_Mant / Cycle: G840 (see page 351)







Contour form

FK see page 60
NS Starting block no. of contour
NE End block no. of contour
X1 Milling top edge (diameter value)
P2 Depth of contour (radius value)

Cycle form

JK Cutter position

- 0: On the contour
- 1, closed contour: Within the contour
- 1, open contour: Left of the contour
- 2, closed contour: Outside the contour
- 2, open contour: Right of the contour
- 3: Depending on H and MD

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed
I Contour-parallel oversize
K Infeed-direction oversize
FZ Infeed rate
E Reduced feed rate
R Approach radius
O Plunging behavior

- 0: Straight (vertical plunge)—The cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour.
- 1: In predrilling—The cycle positions the tool above the hole; the tool plunges and mills the contour.

NF Position mark (only if O=1)
RB Return plane (diameter value)

Further forms: see page 58

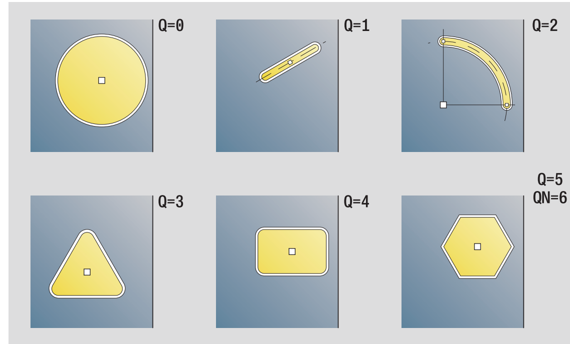
Access to the technology database:

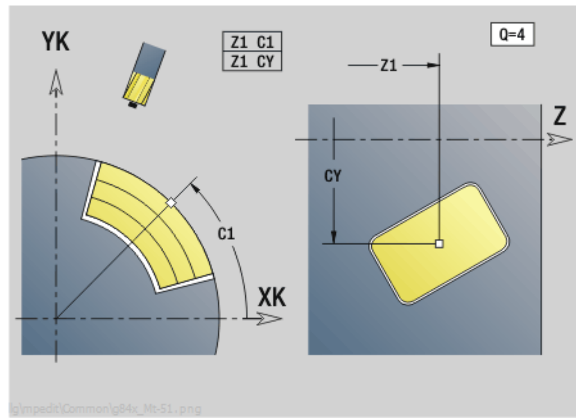
- Machining operation: Finish-milling
- Affected parameters: F, S, FZ, P

"Pocket milling, figures, lateral surface" unit

The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.

Unit name: G84x_Fig_Mant_C / Cycles: G845 (see page 360); G846 (see page 364)





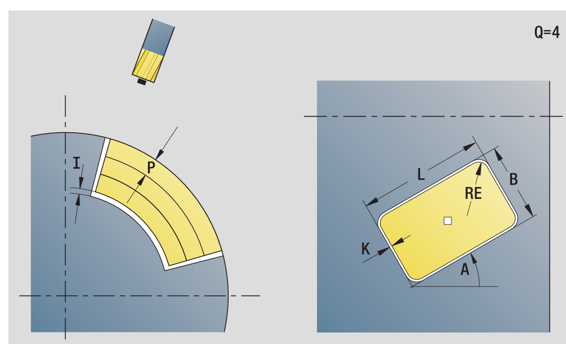


Figure form

Q	Type of figure ■ 0: Full circle ■ 1: Linear slot ■ 2: Circular slot ■ 3: Triangle ■ 4: Rectangle, square ■ 5: Polygon
QN	Number of polygon corners—only with Q=5 (polygon)
Z1	Figure center
C1	Angle of figure center
CY	Figure center of unrolled lateral surface
X1	Milling top edge
P2	Depth of figure
L	Edge length / width across flats ■ L>0: Edge length ■ L<0: Width across flats (inside diameter) for polygon
B	Rectangle width
RE	Rounding radius
A	Angle to Z axis
Q2	Rotational direction of slot—only if Q=2 (circular slot) ■ cw: In clockwise direction ■ ccw: In counterclockwise direction
W	Angle of slot end point—only if Q=2 (circular slot)

Program only the parameters relevant to the selected figure type.

Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

Cycle form

QK Machining operation and plunging strategy

- 0: Roughing
- 1: Finishing
- 2: Helical roughing, manual
- 3: Helical roughing, automatic
- 4: Reciprocating linear roughing, manual
- 5: Reciprocating linear roughing, automatic
- 6: Reciprocating circular roughing, manual
- 7: Reciprocating circular roughing, automatic
- 8: Plunge roughing at predrilling position
- 9: Finishing with 3-D approach arc

JT Machining direction:

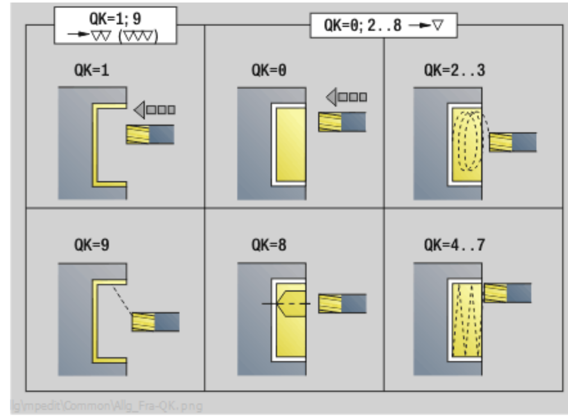
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

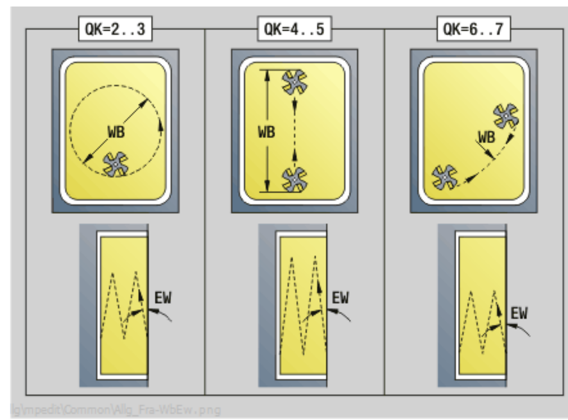
H Cutting direction

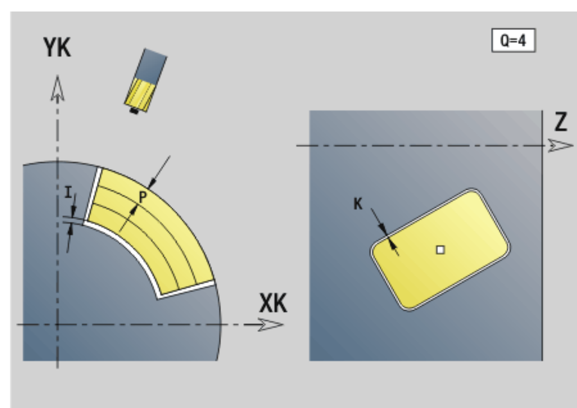
- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed
I Infeed-direction oversize
K Contour-parallel oversize
FZ Infeed rate
E Reduced feed rate
R Approach radius
WB Plunging length
EW Plunging angle
NF Position mark (only if QK=8)
U Overlap factor (default: 0.5)

Further forms: see page 58







"ICP pocket milling, lateral surface" unit

The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.

Unit name: G845_Tas_C_Mant / Cycles: G845 (see page 360); G846 (see page 364)

Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)
P2	Depth of contour
NF	Position mark (only if QK=8)

Cycle form

QK Machining operation and plunging strategy

- 0: Roughing
- 1: Finishing
- 2: Helical roughing, manual
- 3: Helical roughing, automatic
- 4: Reciprocating linear roughing, manual
- 5: Reciprocating linear roughing, automatic
- 6: Reciprocating circular roughing, manual
- 7: Reciprocating circular roughing, automatic
- 8: Plunge roughing at predrilling position
- 9: Finishing with 3-D approach arc

JT Machining direction

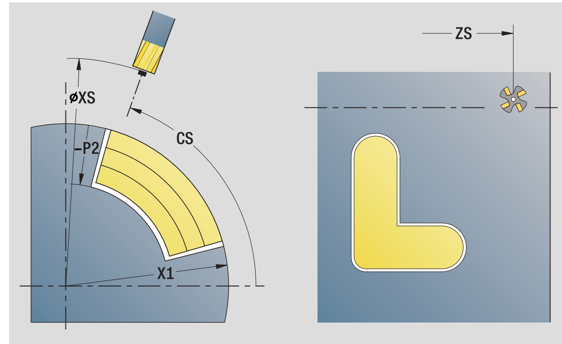
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

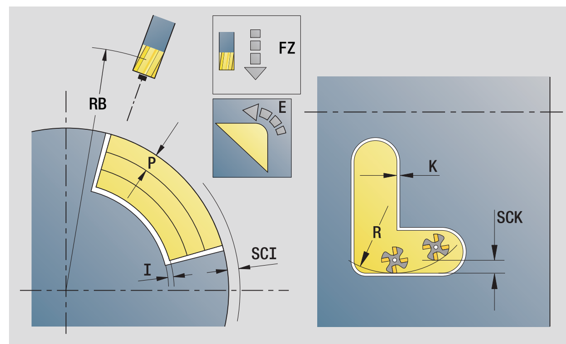
H Cutting direction

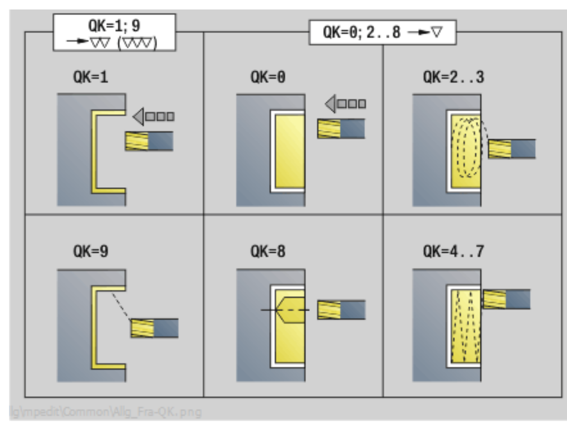
- 0: Up-cut milling
- 1: Climb milling

P	Maximum infeed
I	Infeed-direction oversize
K	Contour-parallel oversize
FZ	Infeed factor
E	Reduced feed rate
R	Approach radius
WB	Plunging length
EW	Plunging angle
U	Overlap factor (default: 0.5)
RB	Return plane (diameter value)

Further forms: see page 58







Access to the technology database:

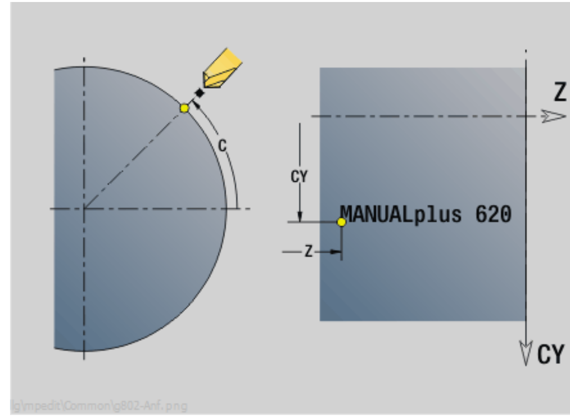
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Engraving, lateral surface" unit

The unit engraves character strings aligned linearly on the lateral surface. Diacritics and special characters that you cannot enter in the smart.Turn editor can be defined, character by character, in **NF**. If you program "Continue from last text" (Q=1), tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.

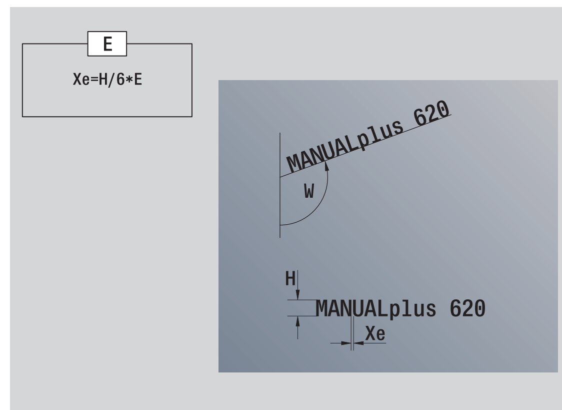
Unit name: G802_GRA_MANT_C / Cycle: G802 (see page 369)

Character set: see page 366



Position form

Z	Start point
C	Starting angle
CY	Start point
X	Final point (diameter). X position, infeed depth during milling.
RB	Return plane

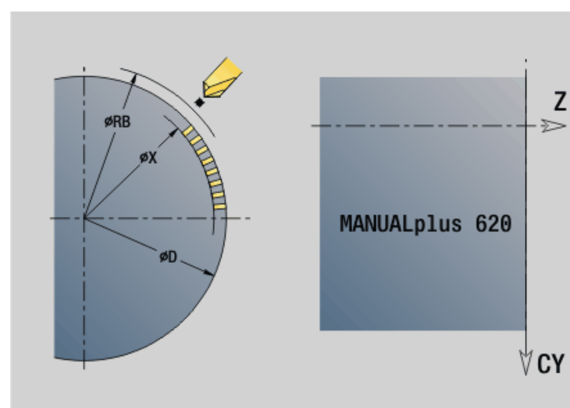


Cycle form

TXT	Text to be engraved
NF	Character number (character to be engraved)
H	Font height
E	Distance factor (for calculation see figure)
W	Inclination angle
FZ	Plunging feed rate factor (plunging feed rate = current feed rate * FZ)
D	Reference diameter
Q	Continue from last text

- 0 (No): Engraving starts at the starting point
- 1 (Yes): Engraving starts at the tool position

Further forms: see page 58



Access to the technology database:

- Machining operation: Engraving
- Affected parameters: F, S

"Deburring, lateral surface" unit

The unit deburrs the contour defined with ICP on the lateral surface.

Unit name: G840_ENT_C_MANT / Cycle: G840 (see page 355)

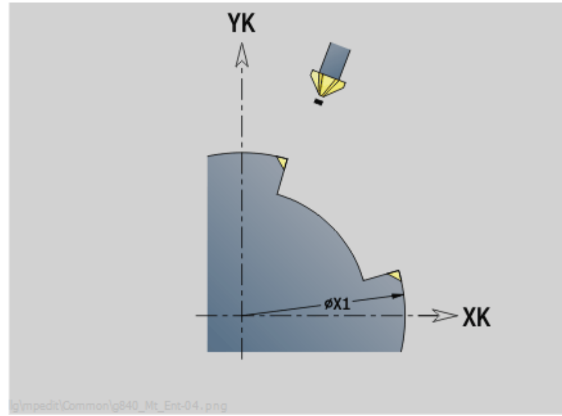
Contour form

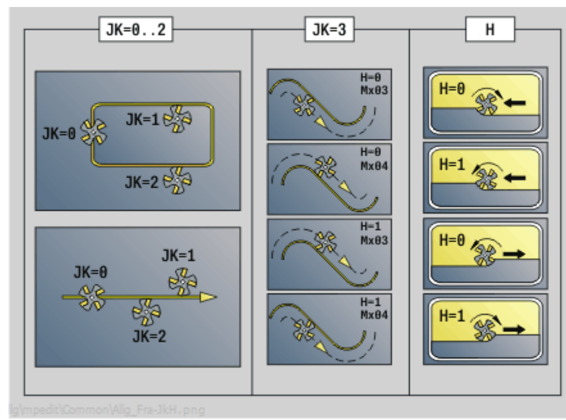
FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)

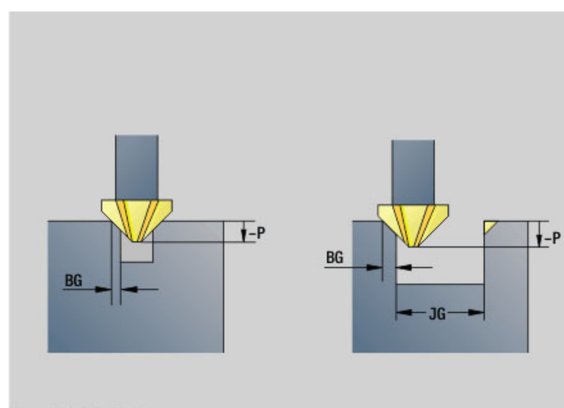
Cycle form

JK	Cutter position ■ JK=0: On the contour ■ JK=1, closed contour: Within the contour ■ JK=1, open contour: Left of the contour ■ JK=2, closed contour: Outside the contour ■ JK=2, open contour: Right of the contour ■ JK=3: Depending on H and MD
H	Cutting direction ■ 0: Up-cut milling ■ 1: Climb milling
BG	Chamfer width
JG	Preparation diameter
P	Plunging depth (indicated as a negative value)
K	Contour-parallel oversize
R	Approach radius
FZ	Infeed rate
E	Reduced feed rate
RB	Return plane

Further forms: see page 58







Access to the technology database:

- Machining operation: Deburring
- Affected parameters: F, S

2.11 Units—Special operations

"Program beginning (START)" unit

In the start unit, default values that are used in the following units are defined. The start unit is called once at the beginning of the machining section. You also directly specify the rotational speed limits, zero point shift and tool change point for the program.

Unit name: Start / Called cycle: None

"Limits" form

S0	Maximum main spindle speed
S1	Maximum rotational speed of driven tool
Z	Zero point shift (G59)

"TC point" form (tool change point)

WT1 Tool change point

- No axis (do not approach the tool change point)
- 0: Simultaneous (X and Z axes depart diagonally)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Only Y
- 6: Simultaneous with Y

WX1	Tool change point in X (reference: distance of the slide position as radius value from the machine zero point)
WZ1	Tool change point in Z (reference: distance of the slide position from the machine zero point)
WY1	Tool change point in Y (reference: distance of the slide position from the machine zero point)

Soft keys in the program beginning form	
Acceptance of zero pnt	Loads the zero point defined during setup
Acceptance TC point $1	Loads the tool-change point defined during setup

"Defaults" form

GWW Tool change point

- No axis (do not approach the tool change point)
- 0: Simultaneous (X and Z axes depart diagonally)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Only Y
- 6: Simultaneous with Y

CLT Coolant

- 0: Without
- 1: Circuit 1 on
- 2: Circuit 2 on

G60 Protection zone: (default for drilling units)

- 0: Active
- 1: Inactive

"Global" form

G47 Safety clearance

SCK Safety clearance in infeed direction (drilling and milling)

SCI Safety clearance in the working plane (milling)

I, K Oversize in X, Z direction (X: diameter value)

You can load the zero point shift and the tool change point by soft key (see soft-key table).

- The setting in the "**`TC point`**" form applies only within the current program.
- Position of tool change point (WX1, WZ1, WY1):
  - If the tool change point is defined, you use G14 to move to this position.
  - If the tool change point is not defined, you use G14 to move to the position defined in manual mode.

"C axis ON" unit

The unit activates the SPI (spindle) C axis.

Unit name: C_Axis_ON / Called cycle: None

"C axis ON" form

SPI	Workpiece spindle number (0 to 3). Spindle that rotates the workpiece.
C	Approach position

"C axis OFF" unit

The unit deactivates the SPI (spindle) C axis.

Unit name: C_Axis_OFF / Called cycle: None

"C axis OFF" form

SPI	Workpiece spindle number (0 to 3). Spindle that rotates the workpiece.

"Subprogram call" unit

The unit calls the subprogram defined in "L".

Unit name: SUBPROG / Called cycle: Any subprogram

Contour form

L	Subprogram name
Q	Number of repetitions
LA-LF	Transfer values
LH	Transfer value
LN	Transfer value—reference to a block number as contour reference. Is updated during block numbering.

Cycle form

LI-LK	Transfer values
LO	Transfer value
LP	Transfer value
LR	Transfer value
LS	Transfer value
LU	Transfer value
LW-LZ	Transfer values

Further forms: see page 58

- The tool call is not an obligatory parameter in this unit!
- Instead of the text "transfer value," texts can be displayed that were defined in the subprogram. You can also define help graphics for every line of the subprogram (see page 404).

Access to the technology database:

- **Not** possible

"Program section repeat" unit

Use the **Repeat** unit to program a program section repeat. The unit consists of two inseparable parts. Program the unit with the Begin form immediately before the repeatable part, and the unit with the End form immediately behind the repeatable part. Be sure to use the same variable number here.

Unit name: REPEAT / Called cycle: None

"Start" form

AE	Repetition ■ 0: Beginning ■ 1: End
V	Variable number 1–30 (counting variable for the iteration loop)
NN	Number of repetitions
QR	Save workpiece blank ■ 0: No ■ 1: Yes
K	Comment

"End" form

AE	Repetition: ■ 0: Beginning ■ 1: End
V	Variable number 1–30 (counting variable for the iteration loop)
Z	Additive datum shift
C	Incremental shift, C axis
Q	Number of the C axis
K	Comment

"Program end" unit

In every smart.Turn program, the end unit should be called once at the end of the machining section.

Unit name: END / Called cycle: None

"Program end" form

ME Type of return jump

- 30: Without M30 restart
- 99: With M99 restart

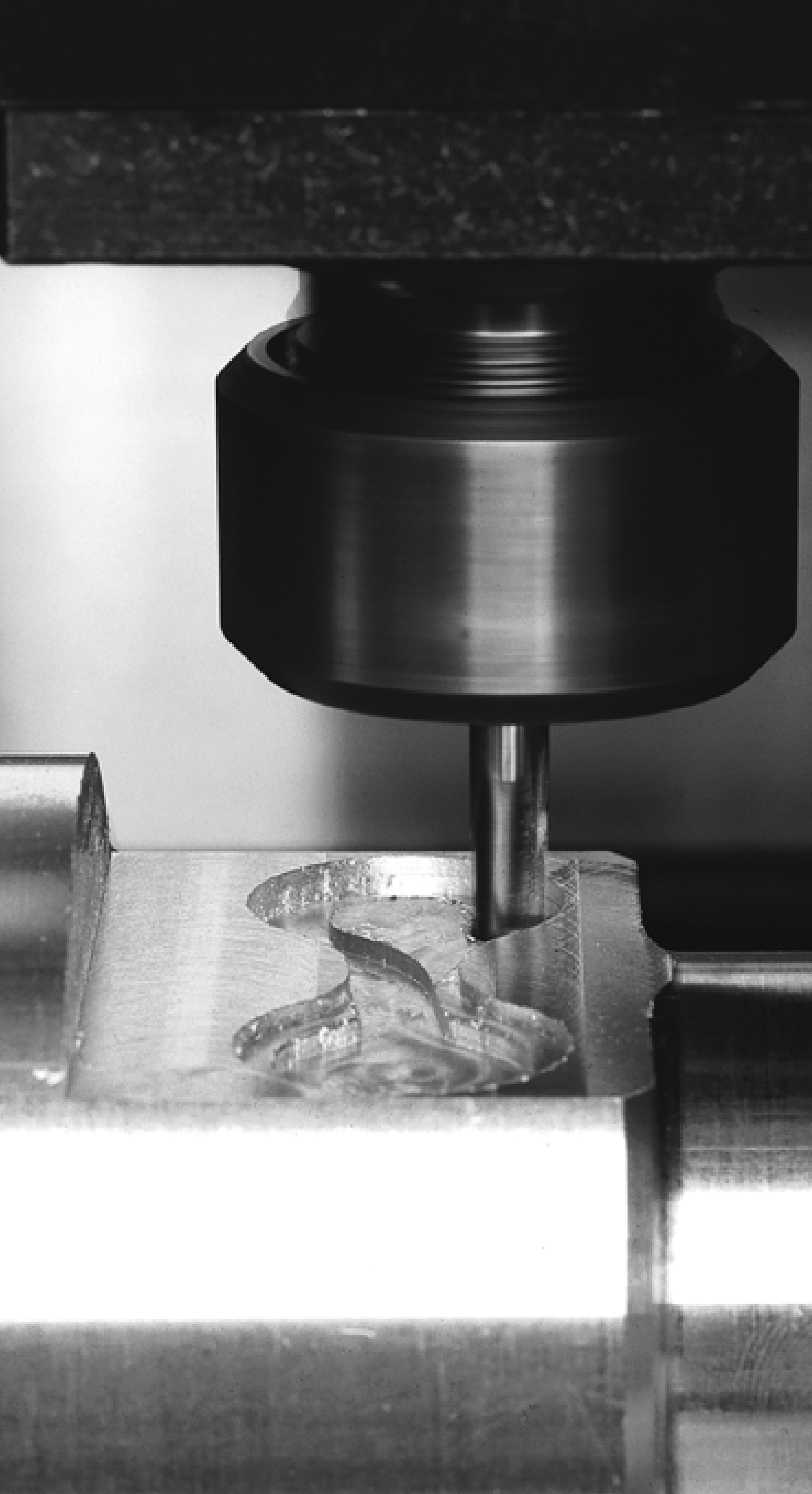
NS Block number for return jump

G14 Tool change point

- No axis (do not approach the tool change point)
- 0: Simultaneous (X and Z axes depart diagonally)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Only Y
- 6: Simultaneous with Y

MFS M command at the start of the unit

MFE M command at the end of the unit



3

smart.Turn units for the Y axis

3.1 Units—Drilling in the Y axis

"ICP drilling, Y axis" unit

The unit machines a single hole or a hole pattern in the XY or YZ plane. Using ICP, you define the holes as well as further details.

Unit name: G74_ICP_Y / Cycle: G74 (see page 321)

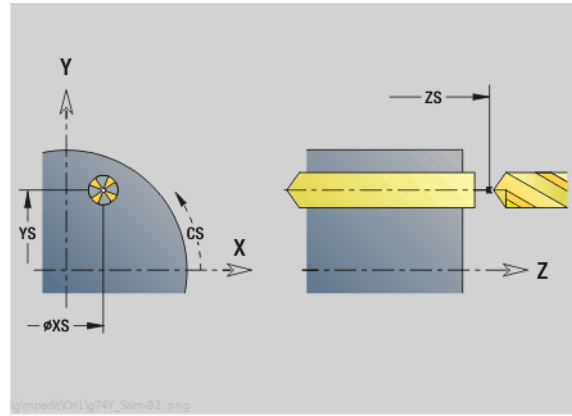
Parameters on the Pattern form

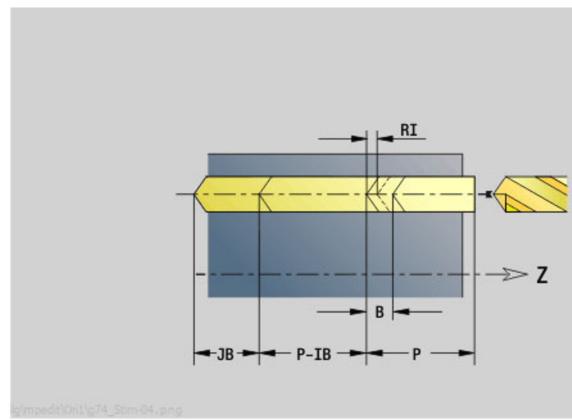
FK	see page 60
NS	Starting block no. of contour

Parameters on the Cycle form

E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
V	Feed rate reduction ■ 0: Without reduction ■ 1: At end of the hole ■ 2: At start of the hole ■ 3: At start and end of the hole
AB	Spot drilling / through drilling length (distance for feed rate reduction)
P	First hole depth
IB	Hole depth reduction value
JB	Minimum hole depth
B	Retraction distance
RI	Internal safety clearance: Distance for reapproach inside the hole (default: safety clearance SCK).
RB	Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58





Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

"ICP tapping, Y axis" unit

The unit machines a single tap hole or a hole pattern in the XY or YZ plane. Using ICP, you define the tap holes as well as further details.

Unit name: G73_ICP_Y / Cycle: G73 (see page 318)

Parameters on the Pattern form

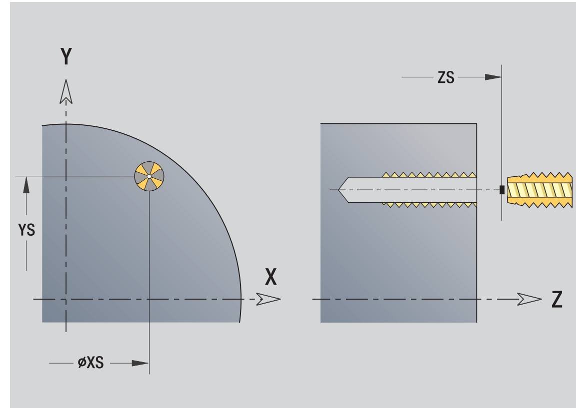
FK	see page 60
NS	Starting block no. of contour

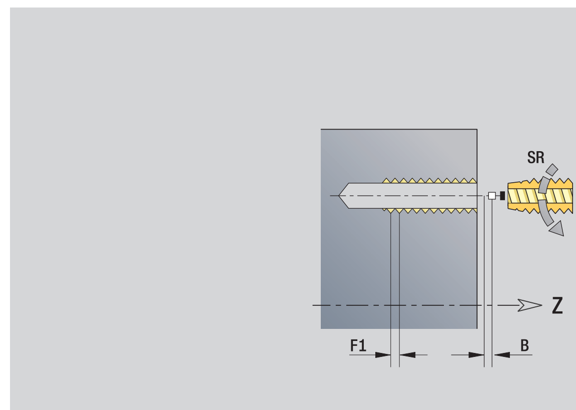
Parameters on the Cycle form

F1	Thread pitch
B	Run-in length
L	Retraction length when using floating tap holders (default: 0)
SR	Retraction speed (default: Shaft speed for tapping)
RB	Return plane

Further forms: see page 58

Retraction length L: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retraction length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from the taps.





Access to the technology database:

- Machining operation: Tapping
- Affected parameter: S

"ICP boring/countersinking, Y axis" unit

The unit machines a single hole or a hole pattern in the XY or YZ plane. Using ICP, you define the hole positions as well as further details for boring or countersinking.

Unit name: G72_ICP_Y / Cycle: G72 (see page 317)

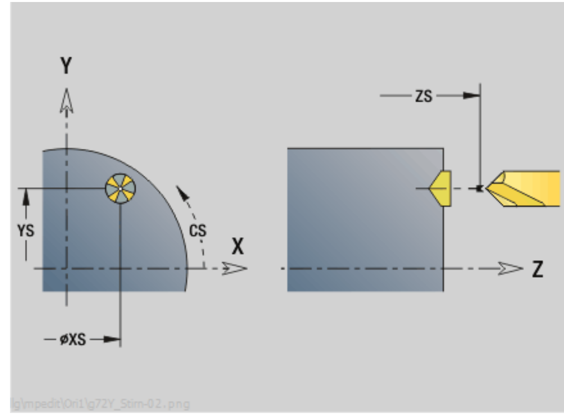
Parameters on the Pattern form

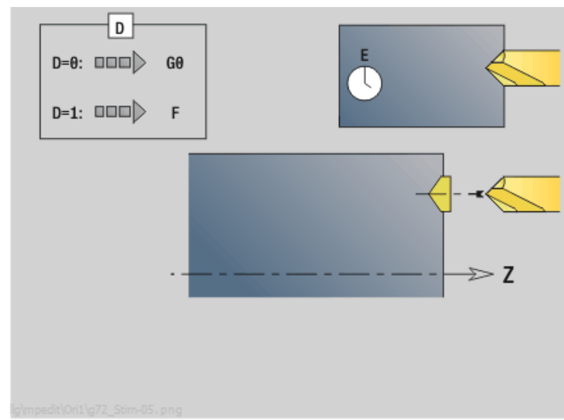
FK	see page 60
NS	Starting block no. of contour

Parameters on the Cycle form

E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
RB	Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58





Access to the technology database:

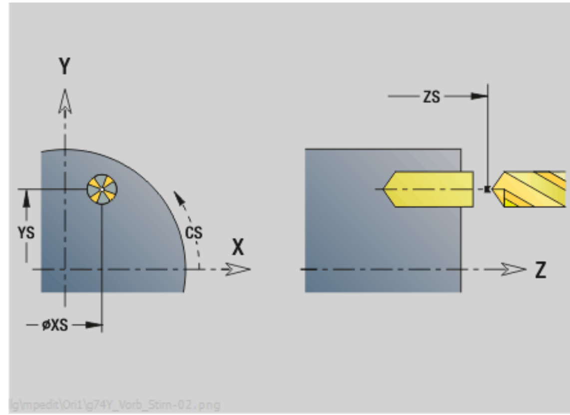
- Machining operation: Drilling
- Affected parameters: F, S

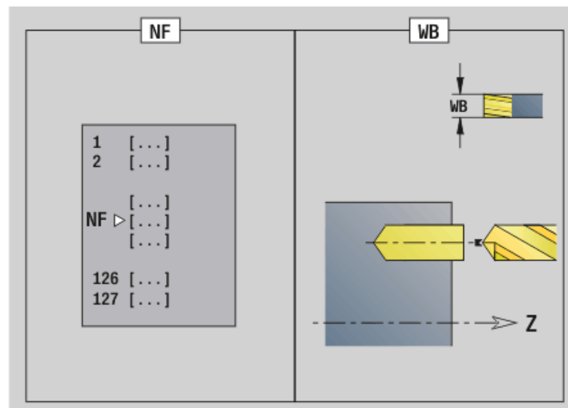
3.2 Units—Predrilling in Y axis

"Predrill, contour mill, ICP in XY plane" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the milling contour consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_STI_840_Y / Cycles: G840 A1 (see page 349); G71 (see page 315)





Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
Z1	Milling top edge
P2	Depth of contour

Parameters on the Cycle form

JK Cutter position

- 0: On the contour
- 1, closed contour: Within the contour
- 1, open contour: Left of the contour
- 2, closed contour: Outside the contour
- 2, open contour: Right of the contour
- 3: Depending on H and MD

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

I	Contour-parallel oversize
K	Infeed-direction oversize
R	Approach radius
WB	Cutter diameter
NF	Position mark
E	Delay (dwell time at end of hole) (default: 0)

D Retraction at

- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction

- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB	Spot drilling / through drilling length (distance for feed rate reduction)
RB	Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58

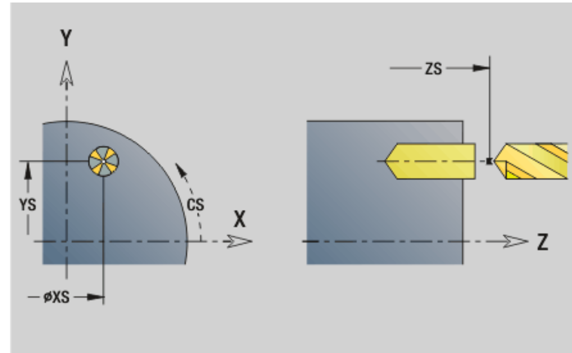
Access to the technology database:

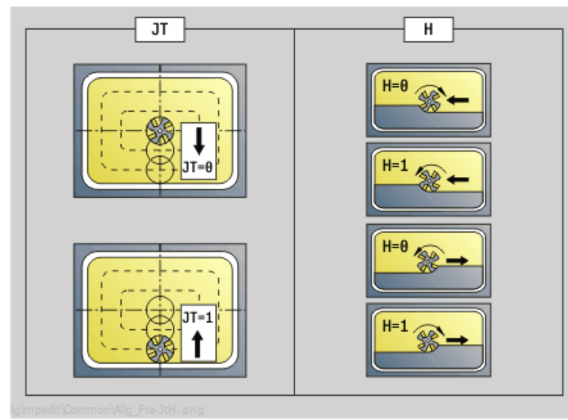
- Machining operation: Drilling
- Affected parameters: F, S

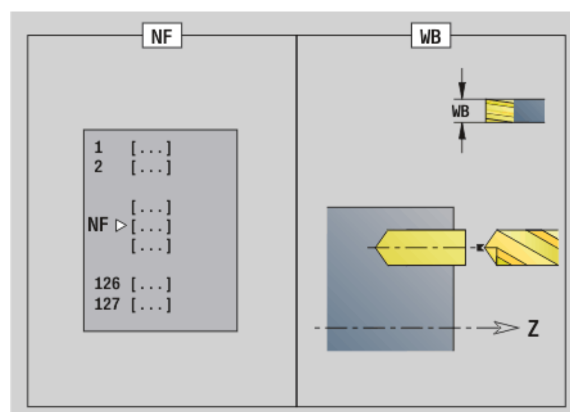
"Predrill, pocket mill, ICP in XY plane" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the pocket consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_STI_845_Y / Cycles: G845 A1 (see page 359); G71 (see page 315)







Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
Z1	Milling top edge
P2	Depth of contour

Parameters on the Cycle form

JT — Machining direction:
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H — Cutting direction
- 0: Up-cut milling
- 1: Climb milling

I — Contour-parallel oversize

K — Infeed-direction oversize

U — Overlap factor (default: 0.5)

WB — Cutter diameter

NF — Position mark

E — Delay (dwell time at end of hole) (default: 0)

D — Retraction at
- 0: Rapid traverse
- 1: Feed rate

V — Feed rate reduction
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB — Spot drilling / through drilling length (distance for feed rate reduction)

RB — Return plane (default: return to the starting position or to the safety clearance)

Further forms: see page 58

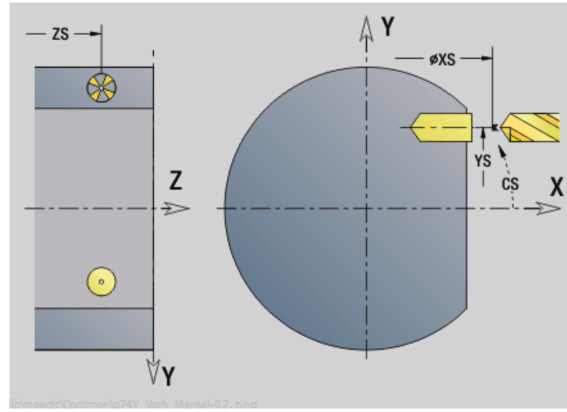
Access to the technology database:

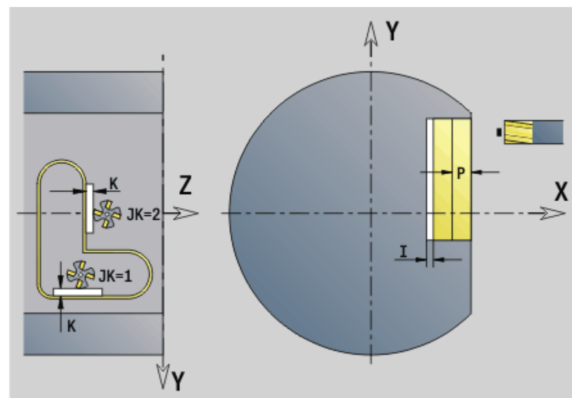
- Machining operation: Drilling
- Affected parameters: F, S

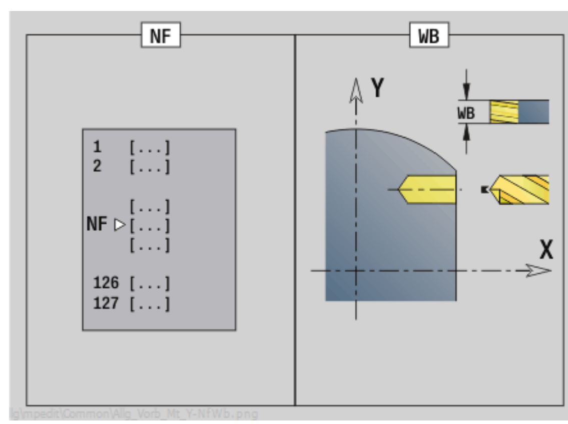
"Predrill, contour mill, ICP in YZ plane" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the milling contour consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_MAN_840_Y / Cycles: G840 A1 (see page 349); G71 (see page 315)







Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)
P2	Depth of contour (radius value)

Parameters on the Cycle form

JK	Cutter position ■ JK=0: On the contour ■ JK=1, closed contour: Within the contour ■ JK=1, open contour: Left of the contour ■ JK=2, closed contour: Outside the contour ■ JK=2, open contour: Right of the contour ■ JK=3: Depending on H and MD
H	Cutting direction ■ 0: Up-cut milling ■ 1: Climb milling
I	Contour-parallel oversize
K	Infeed-direction oversize
R	Approach radius
WB	Cutter diameter
NF	Position mark
E	Delay (dwell time at end of hole) (default: 0)
D	Retraction at ■ 0: Rapid traverse ■ 1: Feed rate
V	Feed rate reduction ■ 0: Without reduction ■ 1: At end of the hole ■ 2: At start of the hole ■ 3: At start and end of the hole
AB	Spot drilling / through drilling length (distance for feed rate reduction)
RB	Return plane (diameter value)

Further forms: see page 58

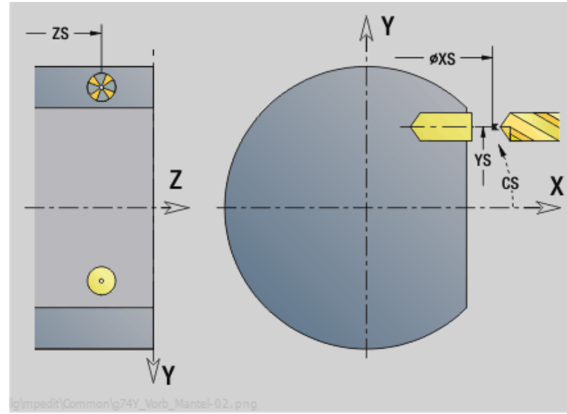
Access to the technology database:

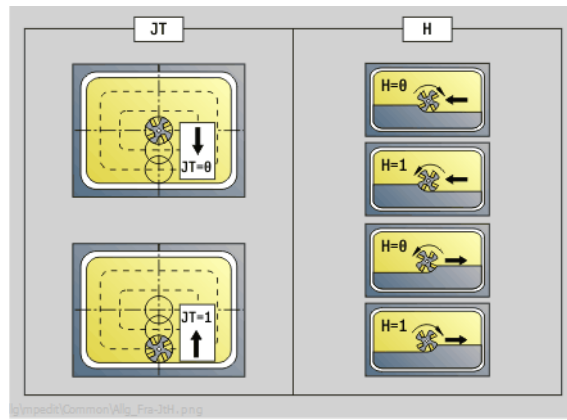
- Machining operation: Drilling
- Affected parameters: F, S

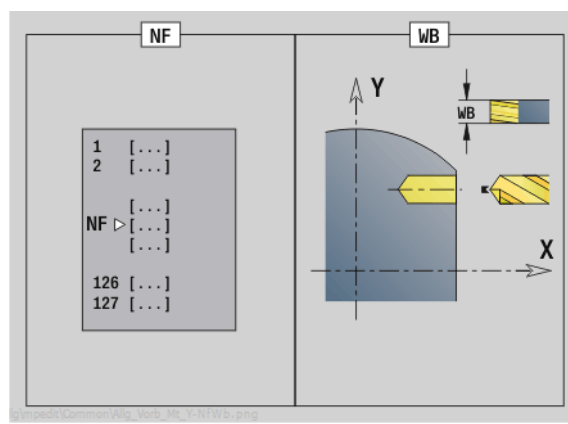
"Predrill, pocket mill, ICP in YZ plane" unit

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in NF. If the pocket consists of multiple sections, the unit machines a hole for each section.

Unit name: DRILL_MAN_845_Y / Cycles: G845 A1 (see page 359)







Parameters on the Contour form

FK see page 60
NS Starting block no. of contour
NE End block no. of contour
X1 Milling top edge (diameter value)
P2 Depth of contour

Parameters on the Cycle form

JT Machining direction:
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

P Maximum infeed
I Infeed-direction oversize
K Contour-parallel oversize
U Overlap factor (default: 0.5)
WB Cutter diameter
NF Position mark
E Delay (dwell time at end of hole) (default: 0)
D Retraction at
- 0: Rapid traverse
- 1: Feed rate

V Feed rate reduction
- 0: Without reduction
- 1: At end of the hole
- 2: At start of the hole
- 3: At start and end of the hole

AB Spot drilling / through drilling length (distance for feed rate reduction)
RB Return plane (diameter value)

Further forms: see page 58

Access to the technology database:

- Machining operation: Drilling
- Affected parameters: F, S

3.3 Units—Milling in Y axis

"ICP contour milling in XY plane" unit

The unit mills the contour defined with ICP in the XY plane.

Unit name: G840_Kon_Y_Stirn / Cycle: G840 (see page 351)

Parameters on the Contour form

FK — see page 60
NS — Starting block no. of contour
NE — End block no. of contour
Z1 — Milling top edge
P2 — Depth of contour

Parameters on the Cycle form

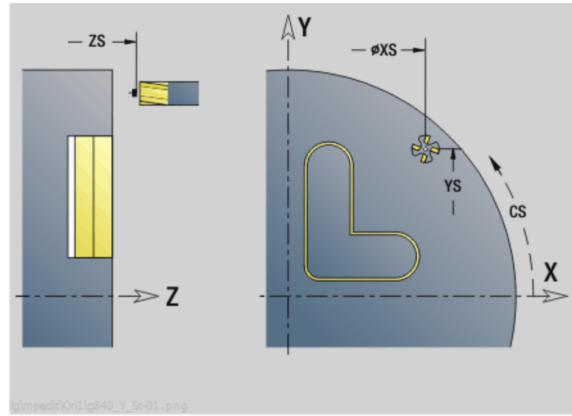
JK — Cutter position
- JK=0: On the contour
- JK=1, closed contour: Within the contour
- JK=1, open contour: Left of the contour
- JK=2, closed contour: Outside the contour
- JK=2, open contour: Right of the contour
- JK=3: Depending on H and MD

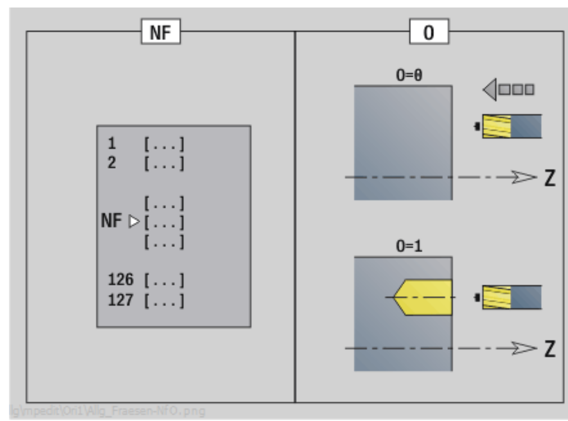
H — Cutting direction
- 0: Up-cut milling
- 1: Climb milling

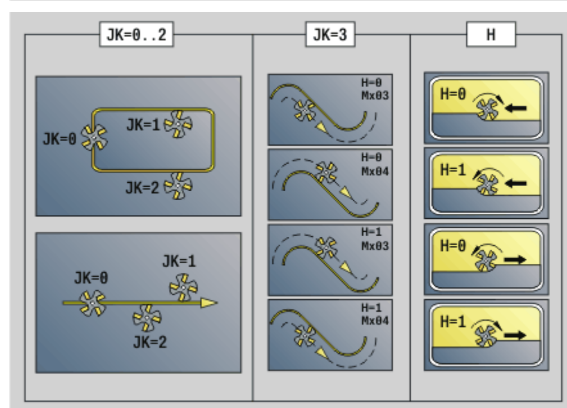
P — Maximum infeed
I — Contour-parallel oversize
K — Infeed-direction oversize
FZ — Infeed rate
E — Reduced feed rate
R — Approach radius
O — Plunging behavior
- 0: Straight (vertical plunge)—The cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour.
- 1: In predrilling—The cycle positions the tool above the hole; the tool plunges and mills the contour.

NF — Position mark (only if O=1)
RB — Return plane

Further forms: see page 58







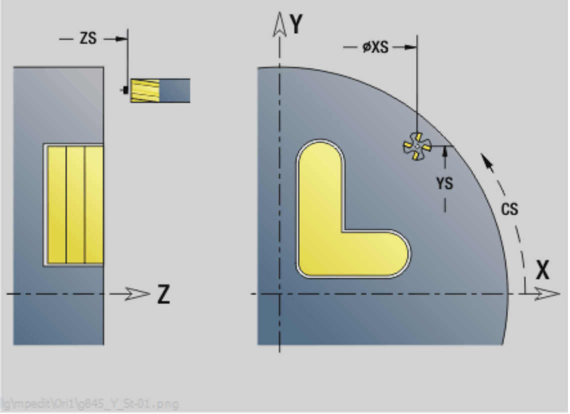
Access to the technology database:

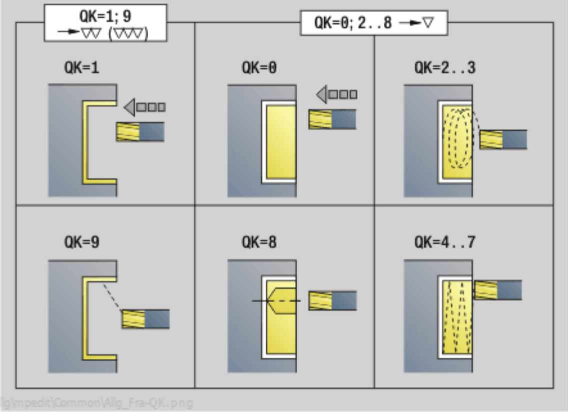
- Machining operation: Finish-milling
- Affected parameters: F, S, FZ, P

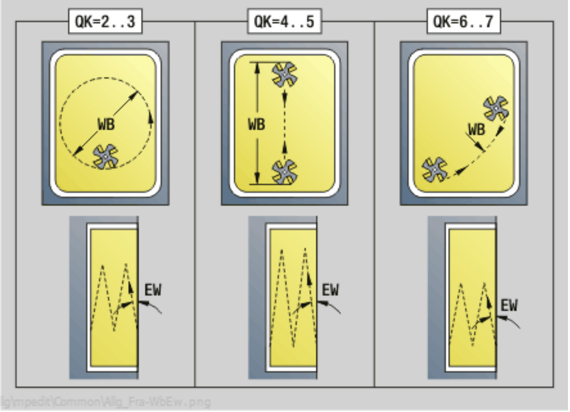
"ICP pocket milling in XY plane" unit

The unit mills the pocket defined with ICP in the XY plane. In **QK** (machining operation), select whether a roughing or finishing operation is to be executed. For roughing, define the plunging strategy.

Unit name: G845_Tas_Y_Stirn / Cycles: G845 (see page 360); G846 (see page 364)







Parameters on the Contour form

FK	see page 60
NF	Position mark (only if QK=8)
NS	Starting block no. of contour
Z1	Milling top edge
P2	Depth of contour
NE	End block no. of contour

Parameters on the Cycle form

QK Machining operation and plunging strategy
- 0: Roughing
- 1: Finishing
- 2: Helical roughing, manual
- 3: Helical roughing, automatic
- 4: Reciprocating linear roughing, manual
- 5: Reciprocating linear roughing, automatic
- 6: Reciprocating circular roughing, manual
- 7: Reciprocating circular roughing, automatic
- 8: Plunge roughing at predrilling position
- 9: Finishing with 3-D approach arc

JT Machining direction:
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

P	Maximum infeed
I	Contour-parallel oversize
K	Infeed-direction oversize
FZ	Infeed rate
E	Reduced feed rate
R	Approach radius
WB	Plunging length
EW	Plunging angle
U	Overlap factor (default: 0.5)
RB	Return plane

Further forms: see page 58

Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Single-surface milling, XY plane" unit

The unit mills a single surface defined with ICP in the XY plane.

Unit name: G841_Y_STI / Cycles: G841 (see page 504); G842 (see page 505)

Parameters on the Contour form

FK see page 60

NS Starting block no. of contour

Parameters on the Cycle form

QK Machining operation:

- 0: Roughing
- 1: Finishing

P Maximum infeed

I Contour-parallel oversize

K Infeed-direction oversize

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

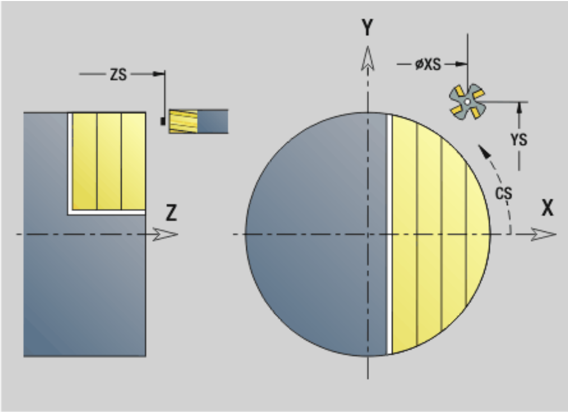
U Overlap factor (default: 0.5)

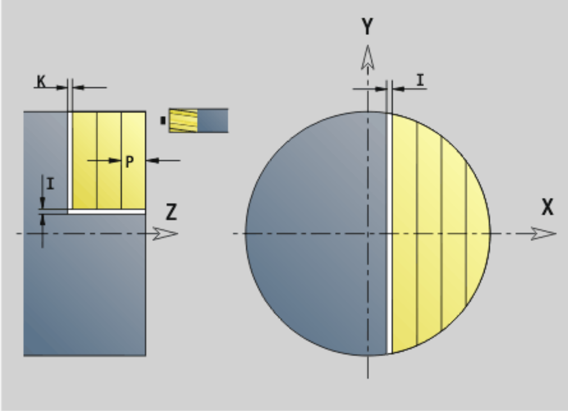
V Overrun factor

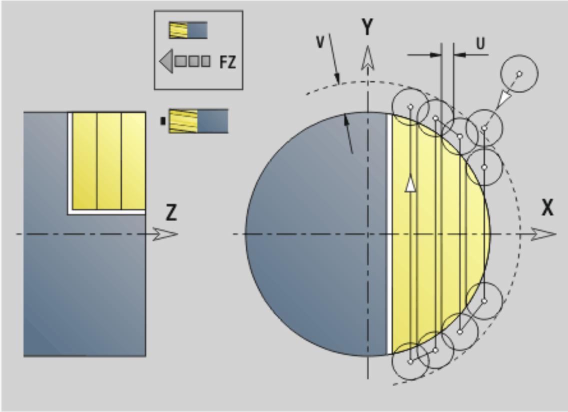
FZ Infeed rate

RB Return plane

Further forms: see page 58







Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Centric polygon milling, XY plane" unit

The unit mills the centric polygon defined with ICP in the XY plane.

Unit name: G843_Y_STI / Cycles: G843 (see page 506); G844 (see page 507)

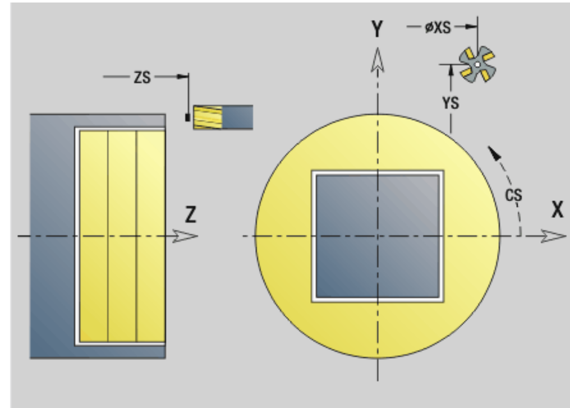
Parameters on the Contour form

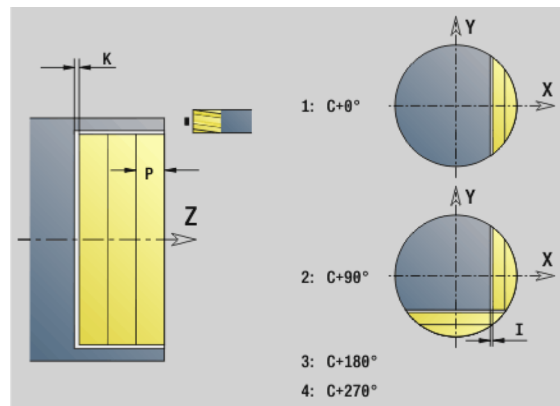
FK	see page 60
NS	Starting block no. of contour

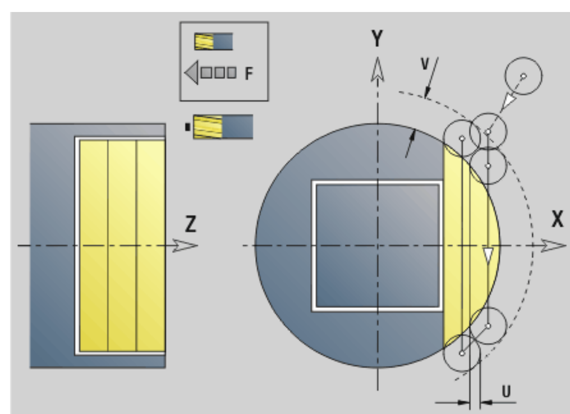
Parameters on the Cycle form

QK	Machining operation: ■ 0: Roughing ■ 1: Finishing
P	Maximum infeed
I	Contour-parallel oversize
K	Infeed-direction oversize
H	Cutting direction ■ 0: Up-cut milling ■ 1: Climb milling
U	Overlap factor (default: 0.5)
V	Overrun factor
FZ	Infeed rate
RB	Return plane

Further forms: see page 58







Access to the technology database:

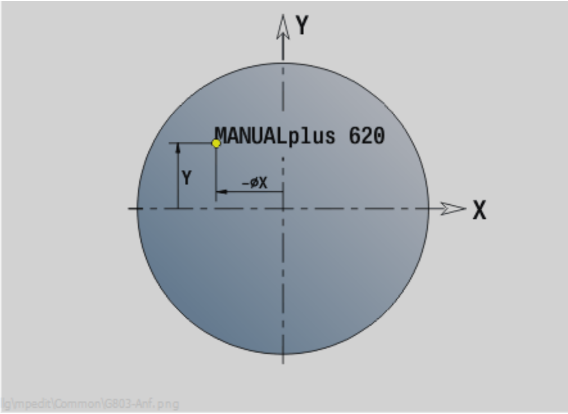
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Engraving in XY plane" unit

The unit engraves character strings aligned linearly in the XY plane. Diacritics and special characters that you cannot enter in the smart.Turn editor can be defined, character by character, in **NF**. If you program "Continue from last text" (Q=1), tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.

Unit name: G803_GRA_Y_STIRN / Cycle: G803 (see page 516)

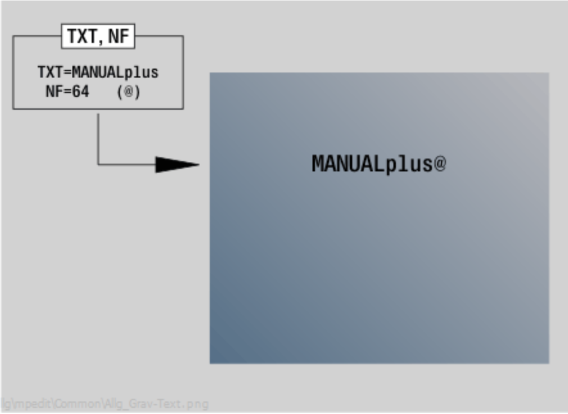
Character set: see page 366



Parameters on the Position form

X, Y	Start point
Z	End point. Z position, infeed depth during milling.
RB	Return plane
APP	Approach: see page 63
DEP	Departure: see page 63

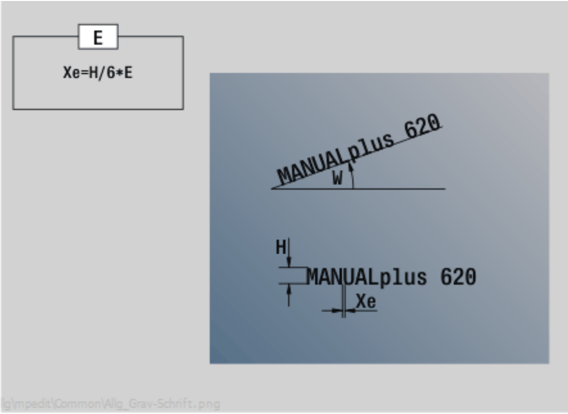
Parameters on the Cycle form



TXT	Text to be engraved
NF	Character number (character to be engraved)
H	Font height
E	Distance factor (for calculation see figure)
W	Inclination angle
FZ	Plunging feed rate factor (plunging feed rate = current feed rate * FZ)
Q	Continue from last text

- 0 (No): Engraving starts at the starting point
- 1 (Yes): Engraving starts at the tool position

Further forms: see page 58



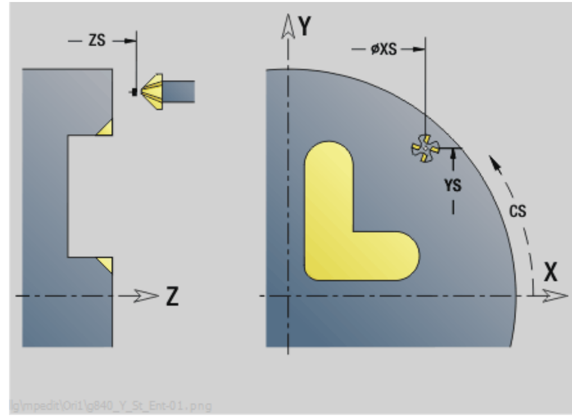
Access to the technology database:

- Machining operation: Engraving
- Affected parameters: F, S

"Deburring in XY plane" unit

The unit deburrs the contour defined with ICP in the XY plane.

Unit name: G840_ENT_Y_STIRN / Cycle: G840 (see page 355)



Parameters on the Contour form

FK see page 60
NS Starting block no. of contour
NE End block no. of contour
Z1 Milling top edge

Parameters on the Cycle form

JK Cutter position

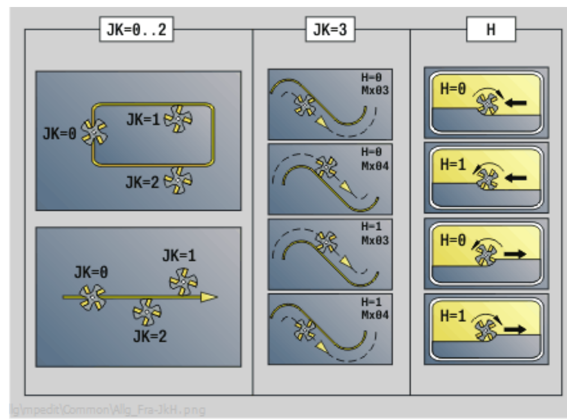
- JK=0: On the contour
- JK=1, closed contour: Within the contour
- JK=1, open contour: Left of the contour
- JK=2, closed contour: Outside the contour
- JK=2, open contour: Right of the contour
- JK=3: Depending on H and MD

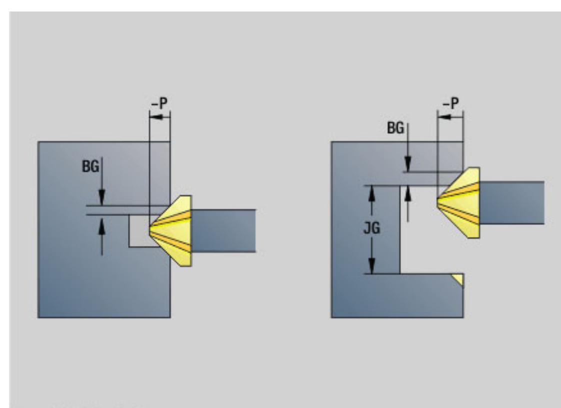
H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

BG Chamfer width
JG Preparation diameter
P Plunging depth (indicated as a negative value)
I Contour-parallel oversize
R Approach radius
FZ Infeed rate
E Reduced feed rate
RB Return plane

Further forms: see page 58





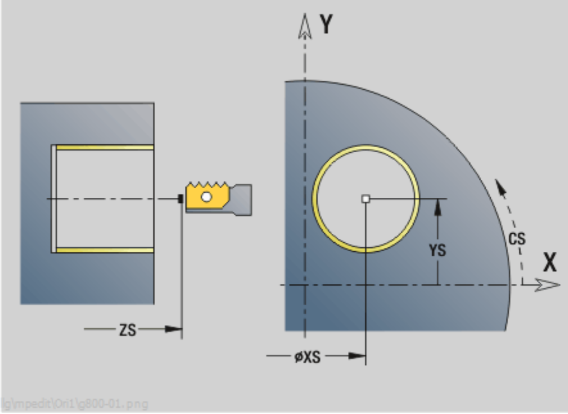
Access to the technology database:

- Machining operation: Deburring
- Affected parameters: F, S

"Thread milling in XY plane" unit

The unit mills a thread in existing holes in the XY plane.

Unit name: G800_GEW_Y_STIRN / Cycle: G800 (see page 518)



Parameters on the Position form

APP Approach see page 63
CS Approach position C
Z1 Start point drill (starting point of hole)
P2 Thread depth
I Thread diameter
F1 Thread pitch

Parameters on the Cycle form

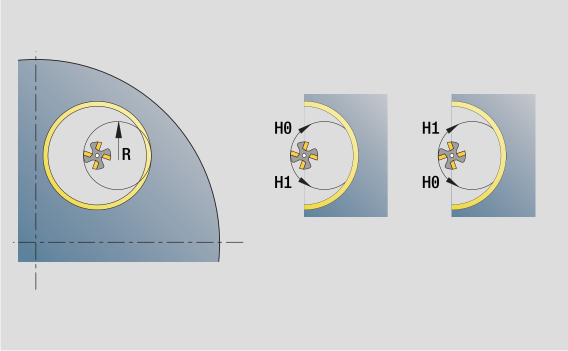
J Direction of thread:
- 0: Right-hand thread
- 1: Left-hand thread

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

V Milling method
- 0: The thread is milled in a 360-degree helix
- 1: The thread is milled in several helical paths (single-point tool)

R Approach radius

Further forms: see page 58



Access to the technology database:
- Machining operation: Finish-milling
- Affected parameters: F, S

"ICP contour milling in YZ plane" unit

The unit mills the contour defined with ICP in the YZ plane.

Unit name: G840_Kon_Y_Mant / Cycle: G840 (see page 351)

Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)
P2	Depth of contour (radius value)

Parameters on the Cycle form

JK Cutter position

- JK=0: On the contour
- JK=1, closed contour: Within the contour
- JK=1, open contour: Left of the contour
- JK=2, closed contour: Outside the contour
- JK=2, open contour: Right of the contour
- JK=3: Depending on H and MD

H Cutting direction

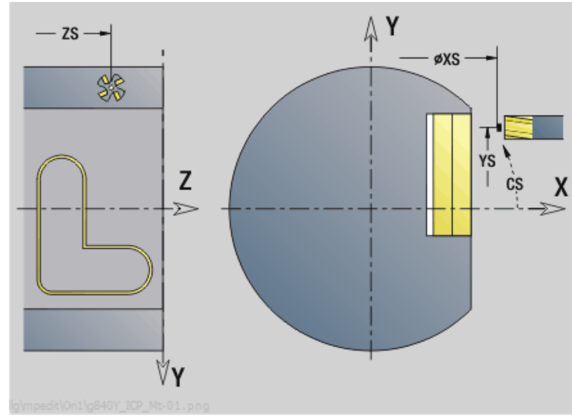
- 0: Up-cut milling
- 1: Climb milling

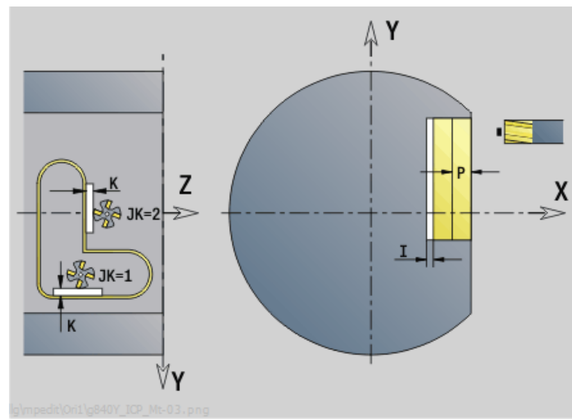
P	Maximum infeed
I	Contour-parallel oversize
K	Infeed-direction oversize
FZ	Infeed rate
E	Reduced feed rate
R	Approach radius
O	Plunging behavior

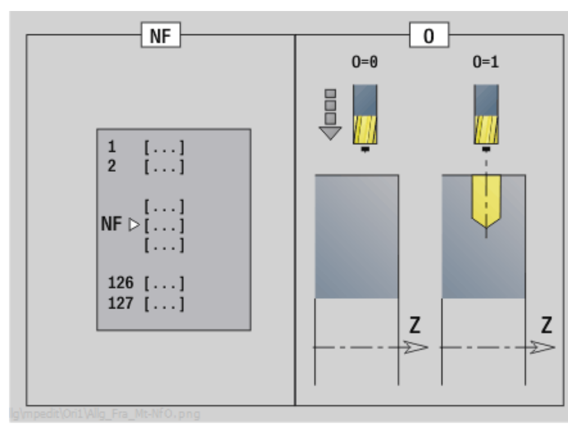
- 0: Straight (vertical plunge)—The cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour.
- 1: In predrilling—The cycle positions the tool above the hole; the tool plunges and mills the contour.

NF	Position mark (only if O=1)
RB	Return plane (diameter value)

Further forms: see page 58







Access to the technology database:

- Machining operation: Finish-milling
- Affected parameters: F, S, FZ, P

"ICP pocket milling in YZ plane" unit

The unit mills the pocket defined with ICP in the YZ plane. In **QK** (machining operation), select whether a roughing or finishing operation is to be executed. For roughing, define the plunging strategy.

Unit name: G845_Tas_Y_Mant / Cycles: G845 (see page 360); G846 (see page 364)

Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)
P2	Depth of contour
NF	Position mark (only if QK=8)

Parameters on the Cycle form

QK Machining operation and plunging strategy

- 0: Roughing
- 1: Finishing
- 2: Helical roughing, manual
- 3: Helical roughing, automatic
- 4: Reciprocating linear roughing, manual
- 5: Reciprocating linear roughing, automatic
- 6: Reciprocating circular roughing, manual
- 7: Reciprocating circular roughing, automatic
- 8: Plunge roughing at predrilling position
- 9: Finishing with 3-D approach arc

JT Machining direction:

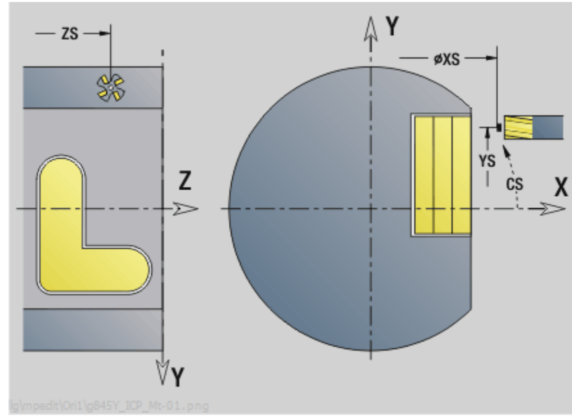
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

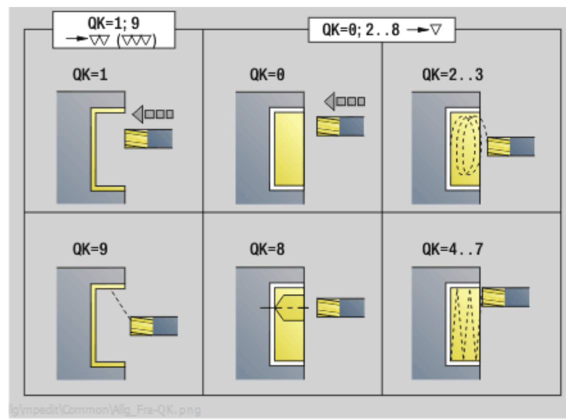
H Cutting direction

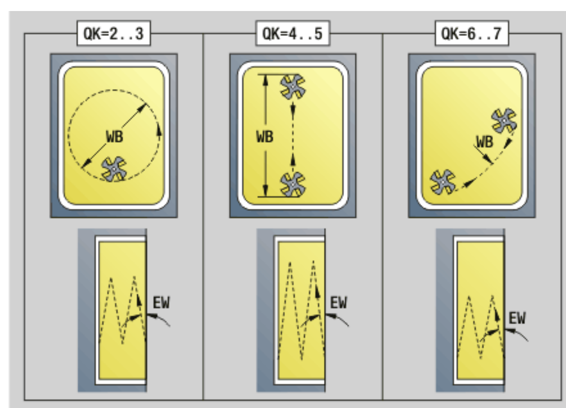
- 0: Up-cut milling
- 1: Climb milling

P	Maximum infeed
I	Infeed-direction oversize
K	Contour-parallel oversize
FZ	Infeed rate
E	Reduced feed rate
R	Approach radius
WB	Plunging length
EW	Plunging angle
U	Overlap factor (default: 0.5)
RB	Return plane (diameter value)

Further forms: see page 58







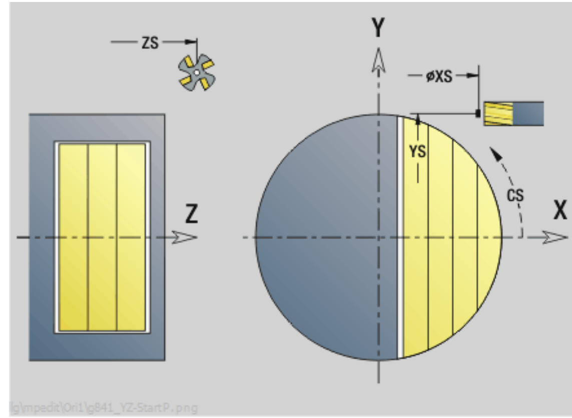
Access to the technology database:

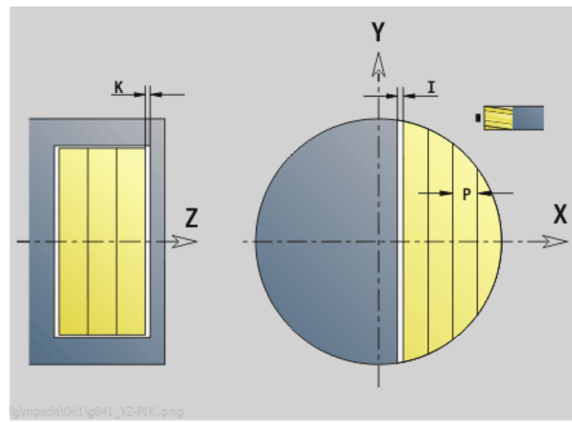
- Machining operation: Milling
- Affected parameters: F, S, FZ, P

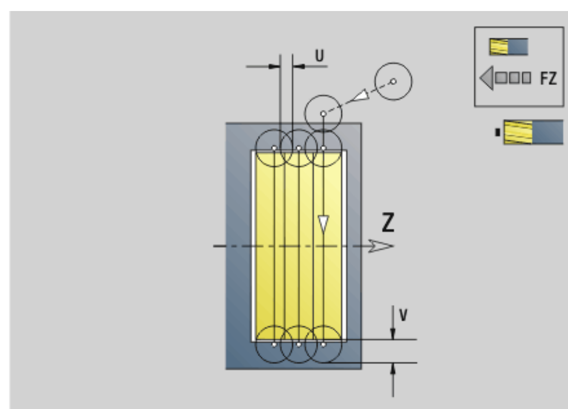
"Single-surface milling, YZ plane" unit

The unit mills a single surface defined with ICP in the YZ plane.

Unit name: G841_Y_MANT / Cycles: G841 (see page 504), G842 (see page 505)







Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour

Parameters on the Cycle form

- QK Machining operation:
  - 0: Roughing
  - 1: Finishing
- P Maximum infeed
- I Contour-parallel oversize
- K Infeed-direction oversize
- H Cutting direction
  - 0: Up-cut milling
  - 1: Climb milling
- U Overlap factor (default: 0.5)
- V Overrun factor
- FZ Infeed rate
- RB Return plane

Further forms: see page 58

Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

"Centric polygon milling, YZ plane" unit

The unit mills the centric polygon defined with ICP in the YZ plane.

Unit name: G843_Y_MANT / Cycles: G843 (see page 506); G844 (see page 506)

Parameters on the Contour form

FK see page 60
NS Starting block no. of contour

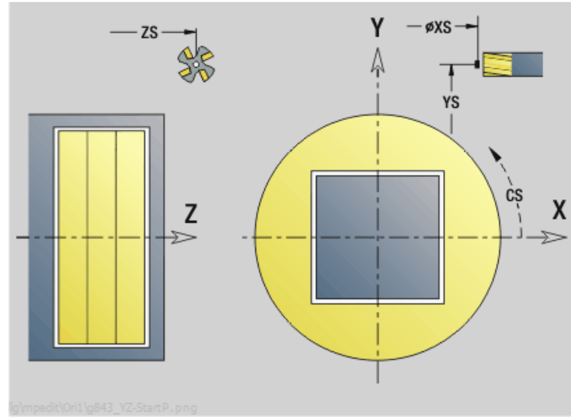
Parameters on the Cycle form

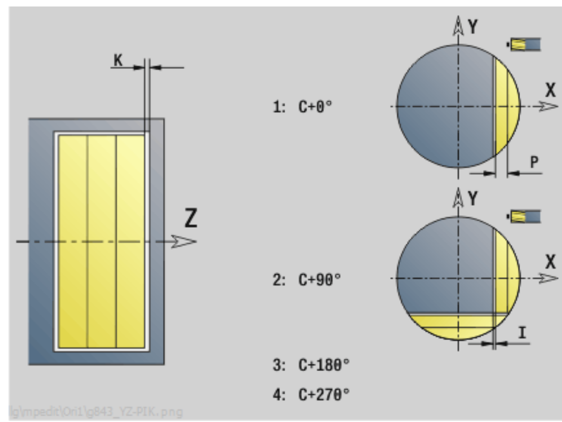
QK Machining operation:
- 0: Roughing
- 1: Finishing

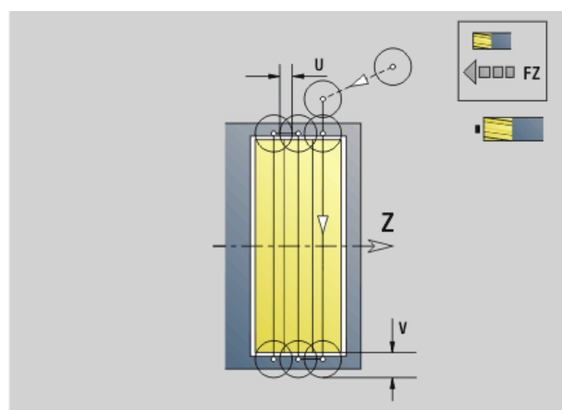
P Maximum infeed
I Contour-parallel oversize
K Infeed-direction oversize
H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

U Overlap factor (default: 0.5)
V Overrun factor
FZ Infeed rate
RB Return plane

Further forms: see page 58





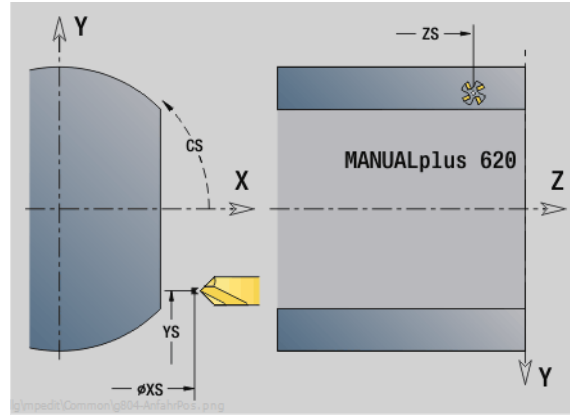


Access to the technology database:

- Machining operation: Milling
- Affected parameters: F, S, FZ, P

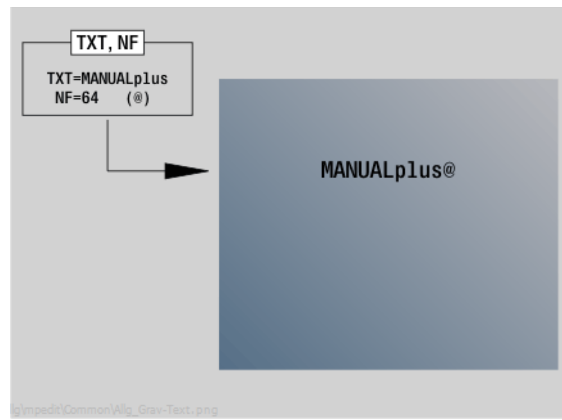
"Engraving in YZ plane" unit

The unit engraves character strings aligned linearly in the YZ plane. Diacritics and special characters that you cannot enter in the smart.Turn editor can be defined, character by character, in **NF**. If you program "Continue from last text" (Q=1), tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.



Unit name: G804_GRA_Y_MANT / Cycle: G804 (see page 517)

Character set: see page 366



Parameters on the Position form

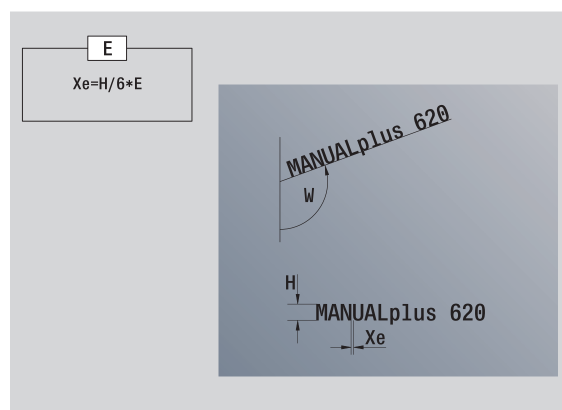
Y, Z	Start point
X	Final point (diameter). X position, infeed depth during milling.
RB	Return plane

Parameters on the Cycle form

TXT	Text to be engraved
NF	Character number (character to be engraved)
H	Font height
E	Distance factor (for calculation see figure)
W	Inclination angle
FZ	Plunging feed rate factor (plunging feed rate = current feed rate * FZ)
Q	Continue from last text

- 0 (No): Engraving starts at the starting point
- 1 (Yes): Engraving starts at the tool position

Further forms: see page 58



Access to the technology database:

- Machining operation: Engraving
- Affected parameters: F, S

"Deburring in YZ plane" unit

The unit deburrs the contour defined with ICP in the YZ plane.

Unit name: G840_ENT_Y_MANT / Cycle: G840 (see page 355)

Parameters on the Contour form

FK	see page 60
NS	Starting block no. of contour
NE	End block no. of contour
X1	Milling top edge (diameter value)

Parameters on the Cycle form

JK Cutter position

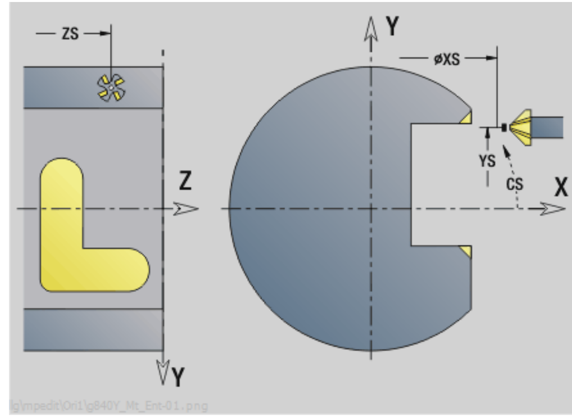
- JK=0: On the contour
- JK=1, closed contour: Within the contour
- JK=1, open contour: Left of the contour
- JK=2, closed contour: Outside the contour
- JK=2, open contour: Right of the contour
- JK=3: Depending on H and MD

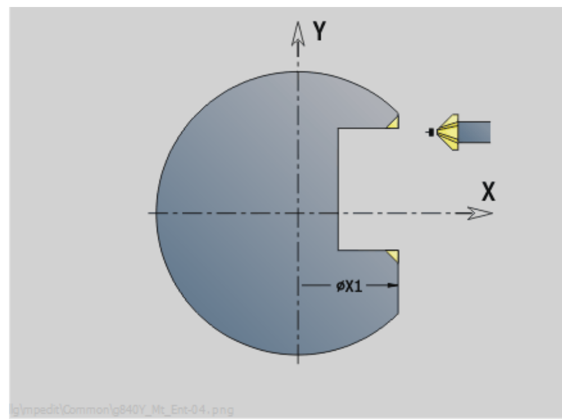
H Cutting direction

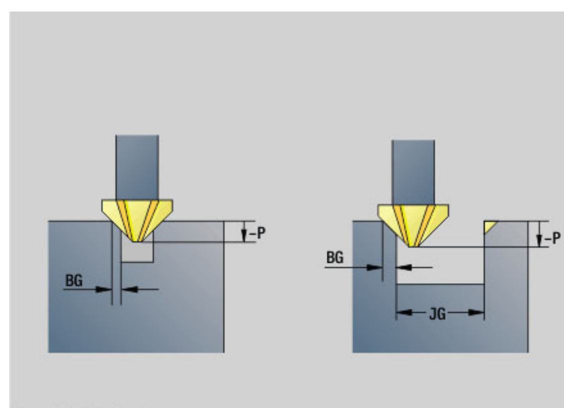
- 0: Up-cut milling
- 1: Climb milling

BG	Chamfer width
JG	Preparation diameter
P	Plunging depth (indicated as a negative value)
K	Contour-parallel oversize
R	Approach radius
FZ	Infeed rate
E	Reduced feed rate
RB	Return plane

Further forms: see page 58







Access to the technology database:

- Machining operation: Deburring
- Affected parameters: F, S

"Thread milling in YZ plane" unit

The unit mills a thread in existing holes in the YZ plane.

Unit name: G806_GEW_Y_MANT / Cycle: G806 (see page 519)

Parameters on the Position form

APP	Approach see page 63
CS	Approach position C
X1	Start point drill (starting point of hole)
P2	Thread depth
I	Thread diameter
F1	Thread pitch

Parameters on the Cycle form

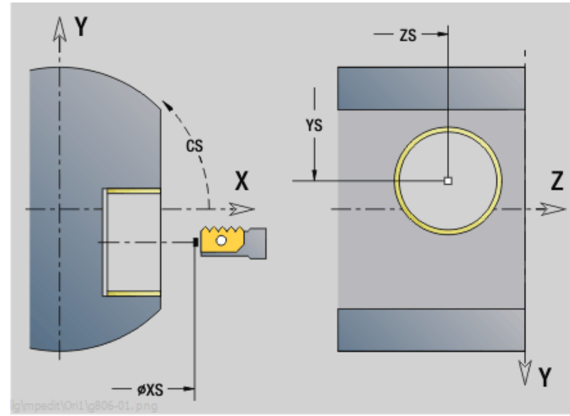
J Direction of thread:
- 0: Right-hand thread
- 1: Left-hand thread

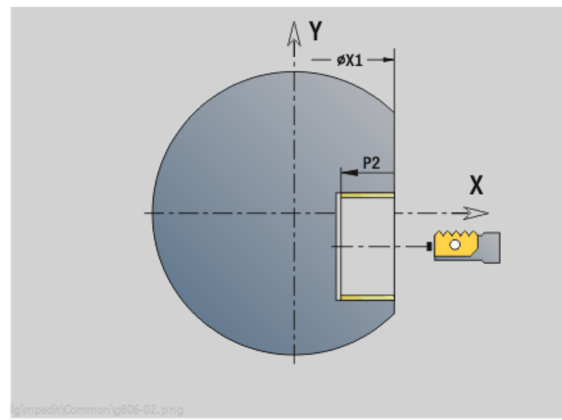
H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

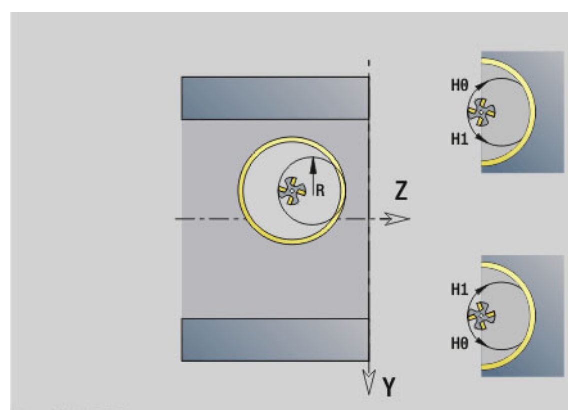
V Milling method
- 0: The thread is milled in a 360-degree helix
- 1: The thread is milled in several helical paths (single-point tool)

R Approach radius

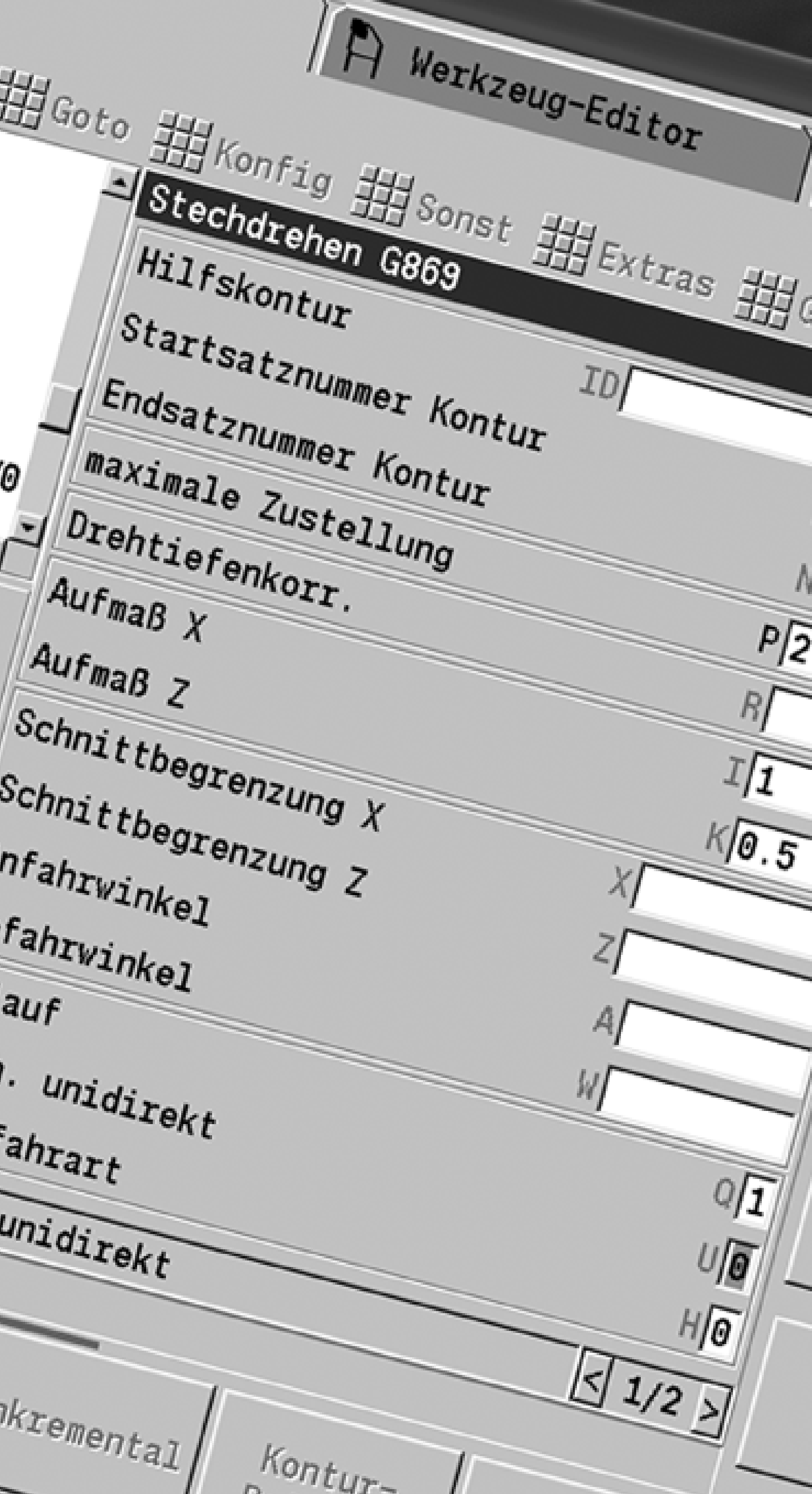
Further forms: see page 58







Access to the technology database:
- Machining operation: Finish-milling
- Affected parameters: F, S



4

DIN programming

4.1 Programming in DIN/ISO mode

Geometry and machining commands

The Control also supports structured programming in DIN/ISO mode.

The **G commands** are divided into:

- **Geometry commands** for describing the blank and finished part.
- **Machining commands** for the MACHINING section.

Some G codes are used for blank/finished-part definition and in the MACHINING section. When copying or shifting NC blocks, keep in mind that "geometry" functions are used exclusively for describing a contour, while "machining" functions are used only in the MACHINING section.

Example: "Structured DINplus program"

HEADER	
#MATERIAL	Steel
#MACHINE	Automatic lathe
#DRAWING	356_787.9
#CLAMP. PRESS.	20
#SLIDE	$1
#COMPANY	Turn & Co
#UNIT	METRIC
TURRET 1	
T1 ID"342-300.1"	
T2 ID"111-80-080.1"	
. . .	
BLANK	
N1 G20 X120 Z120 K2	
FINISHED PART	
N2 G0 X60 Z-115	
N3 G1 Z-105	
. . .	
MACHINING	
N22 G59 Z282	
N25 G14 Q0	
[Predrilling 30 mm outside centric face]	
N26 T1	
N27 G97 S1061 G95 F0.25 M4	
. . .	
END	

Contour programming

The "contour follow-up" function and contour-related turning cycles require the previous description of the blank and finished part. For milling and drilling, contour definition is a precondition if you wish to use fixed cycles.

Use ICP (Interactive Contour Programming) for describing blank and finished parts.

Contours for turning

- Describe a continuous contour.
- The direction of the contour description is independent of the direction of machining.
- Contour descriptions must not extend beyond the turning center.
- The contour of the finished part must lie within the contour of the blank part.
- When machining bars, define only the required section as blank.
- Contour definitions apply to the entire NC program, even if the workpiece is rechucked for machining the rear face.
- In the fixed cycles, the defined contour is used to program "reference values."

To describe **workpiece blanks** and **auxiliary workpiece blanks**, use

- G20 "Blank part macro" for standard parts (cylinder, hollow cylinder).
- G21 "Cast-part macro" for blank-part contours based on finished-part contours. G21 is only used for describing workpiece blanks.
- Individual contour elements (such as are used for finished-part contours) where use of G20 or G21 is not possible.

To describe finished parts, use individual contour elements and form elements. The contour elements or the complete contour can be assigned attributes accounted for during the machining of the workpiece (example: oversizes, additive compensation, special feed rates, etc.). The Control always uses paraxial elements to close finished parts.

For intermediate machining steps, define **auxiliary contours.** Auxiliary contours are programmed in the same way as finished-part descriptions. One contour description is possible per AUXILIARY CONTOUR. An AUXILIARY CONTOUR is assigned a name (ID) that can be referenced by the cycles. Auxiliary contours are not closed automatically.

Contours for C-axis machining:

- Contours for C-axis machining are programmed within the FINISHED PART section.
- Identify the contours as a FRONT or SURFACE. You can use section codes more than once or program multiple contours within one section code.

Block references: When editing G codes related to the contour (MACHINING section), load the block references from the displayed contour.

- Place the cursor in the input box (NS).

Contour reference

- Switch to the contour display.
- Place the cursor on the desired contour element

NE

- Switch to NE.
- Place the cursor on the desired contour element

Take over

- Press the **LOAD** soft key to return to the dialog.

NC blocks of the DIN program

An NC block contains **NC commands** such as positioning, switching or organizational commands. Traversing and switching commands begin with G or M followed by a number (G1, G2, G81, M3, M30, ...) and the address parameters. Organizational commands consist of key words (WHILE, RETURN, etc.), or of a combination of letters/ numbers.

You can also program NC blocks containing only variable calculations.

You can program several NC commands in one NC block, provided they have different address letters and do not have opposing functions.

Examples

- Permissible combination: N10 G1 X100 Z2 M8
- Non-permissible combination:
  N10 G1 X100 Z2 G2 X100 Z2 R30 (same address letters are used more than once) or
  N10 M3 M4 – opposing functionality

NC address parameters
The address parameters consist of 1 or 2 letters followed by

- A value
- A mathematical expression
- A question mark (VGP simplified geometry programming)
- A letter "i" to designate incremental address parameters (examples: Xi..., Ci..., XKi..., YKi..., etc.)
- A **# variable**
- A **constant** (_constname)

Examples:

- X20 [Absolute dimension]
- Zi–35.675 [Incremental dimension]
- X? [Simple geometry programming]
- X#l1 [Variable programming]
- X(#g12+1) [Variable programming]
- X(37+2)*SIN(30) [Mathematical expression]
- X(20*_pi) [Expression with constant]

Creating, editing and deleting NC blocks

Make NC block:

- Press the INS key. The Control creates a new NC block below the cursor position.

- Alternatively you can program the NC command directly. The Control creates a new NC block or inserts the NC command in the existing NC block.

Delete the NC block:

- Position the cursor on the NC block to be deleted.

- Press the DEL key. The Control deletes the NC block.

Add an NC element:

- Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.),
- Insert NC element (G, M, T function, etc.).

Change NC element:

- Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.) or the section code.

- Press ENTER or double-click with the left mouse key. The Control activates a dialog box which displays the block number, the number of the G or M function, or the address parameters, which can then be edited.

Delete NC element:

- Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.),

- Press the DEL key. The NC element highlighted by the cursor **and** all the related elements are deleted. Example: If the cursor is located on a G command, the address parameters are also deleted.

Address parameters

Coordinates can be programmed absolutely or incrementally. If you do not make any entry for X, Y, Z, XK, YK, C, the coordinates of the block previously executed will be retained (modal).

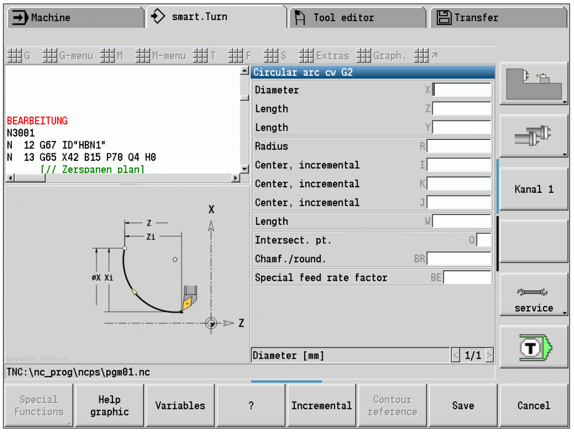
The Control calculates missing coordinates in the principal axes X, Y or Z if you program "?" (simplified geometry programming).

The machining functions G0, G1, G2, G3, G12 and G13 are modal. This means that the Control uses the previous G command if the address parameters X, Y, Z, I or K in the following block have been programmed without a G function However, the address parameters must have been programmed as absolute values.

The Control supports the use of variables and mathematical expressions as address parameters.

To edit address parameters:

- Call the dialog box.
- Position the cursor in the input field and enter/change the values, or
- Use the additional input options provided by the soft keys.
  - "?" Simplified geometry programming
  - Switch from incremental to absolute, or vice versa
  - Activate variable input
  - Load the contour reference.



Soft key in the G dialog	
Help graphic	Alternately shows and hides the help graphics
Variables	Opens the alphabetic keyboard for entering variables (GOTO key)
?	Inserts the question mark for activating the simplified geometry programming
Incremental	Activates incremental programming for the current input parameters
Contour reference	Allows transferring the contour references for NS and NE

Fixed cycles

HEIDENHAIN recommends programming a fixed cycle as follows:

- Insert the tool
- Define the cutting data
- Position the tool in front of the working area
- Define the safety clearance
- Cycle call
- Retract the tool
- Move to tool change position

Danger of collision!

Remember when omitting cycle programming steps during optimization:

- A special feed rate remains in effect until the next feed command (for example the finishing feed rate during recessing cycles).
- Some cycles traverse diagonally back to the starting point if you use the standard programming (for example roughing cycles).

Typical structure of a fixed cycle

. . .	
MACHINING	
N.. G59 Z..	Zero point shift
N.. G26 S..	Define the speed limit
N.. G14 Q..	Move to tool change position
. . .	
N.. T..	Insert the tool
N.. G96 S.. G95 F.. M4	Define the technology data
N.. G0 X.. Z..	Pre-position
N.. G47 P..	Define the safety clearance
N.. G810 NS.. NE..	Cycle call
N.. G0 X.. Z..	If necessary, retract
N.. G14 Q0	Move to tool change position
. . .	

Subprograms, expert programs

Subprograms are used to program the contour or the machining process.

In the subprograms, transfer parameters are available as variables. You can fix the designation of the transfer parameters and illustrate them in help graphics (See "Subroutines" on page 404.).

In every subprogram, the local variables #l1 to #l30 are available for internal calculations.

Subprograms can be nested up to six times. Nesting means that a subprogram calls a further subprogram, etc.

If a subprogram is to be run repeatedly, enter the number of times the subprogram is to be repeated in the Q parameter.

The Control distinguishes between local and external subprograms.

- **Local subprograms** are in the file of the NC main program. Local subprograms can only be called in from their corresponding main programs.
- **External subprograms** are stored in separate NC files and can be called in from any NC main program or other NC subprograms.

Expert programs
An expert program is a subprogram that executes complex processes and is adapted to the machine configurations. Expert programs are usually provided by the machine tool builder.

NC program conversion

For programming and user communication, keep in mind that the Control interprets the NC program up to the fixed word MACHINING in the program selection. The MACHINING section is not interpreted until you select `Cycle on`.

DIN/ISO programs of predecessor controls

The formats of the DIN/ISO programs of the predecessor controls MANUALplus 4110 and CNC PILOT 4290 differ from that of the MANUALplus 620. However, you can use the program converter to adapt programs from the predecessor controls to the new one.

When opening an NC program, the Control recognizes the programs of predecessor controls. The program concerned will be converted after a confirmation prompt. "CONV_..." will be prefixed to the program name.

The converter is also part of the Transfer function (Organization mode of operation).

DIN/ISO programs not only have new solutions for tool management, technology data, etc., but also for contour description and variable programming.

Remember the following when converting **DIN/ISO programs of the MANUALplus 4110**:

- **Tool call:** The loading of the T number depends on whether the program is a "multifix program" (2-digit T number) or "turret program" (4-digit T number).
  - 2-digit T number: The T number is loaded as "ID" and entered as the T number "T1".
  - 4-digit T number (Tddpp): The first two digits of the T number (dd) are loaded as "ID" and the last two (pp) as "T".
- **Workpiece-blank definition:** A G20/G21 workpiece-blank definition of the 4110 becomes an AUXILIARY BLANK.
- **Contour descriptions:** In MANUALplus 4110 programs, the fixed cycles are followed by the contour description. During conversion the contour description is converted to an AUXILIARY CONTOUR. The associated cycle in the MACHINING section then refers to this auxiliary contour.
- **Variable programming:** Variable accesses to tool data, machine dimensions, D compensation values, parameter data and events cannot be converted. These program sequences have to be adapted.
- **M functions** are left unchanged.
- **Inches or metric:** The converter cannot detect the unit of measure of the MANUALplus 4110 program. Consequently, no unit of measure is entered in the target program. This has to be completed by the user.

Remember the following when converting **DIN programs of the CNC PILOT 4290**:

- **Tool call** (T commands of the TURRET section):
  - T commands containing a reference to the tool database are left unchanged (example: T1 ID"342-300.1").
  - T commands containing tool data cannot be converted.
- **Variable programming:** Variable accesses to tool data, machine dimensions, D compensation values, parameter data and events cannot be converted. These program sequences have to be adapted.
- **M functions** are left unchanged.
- **Names of external subprograms:** When an external subprogram is called, the converter prefixes "CONV_..." to the name.

If the DIN program contains nonconvertible elements, the corresponding NC block is saved as a comment. The word WARNING is inserted in front of this comment. Depending on the situation, the nonconvertible command is taken into the comment line, or the nonconvertible NC block follows the comment.

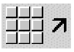
HEIDENHAIN recommends adapting converted NC programs to the circumstances of theControl and then testing them before using them for production.

"Geometry" pull-down menus

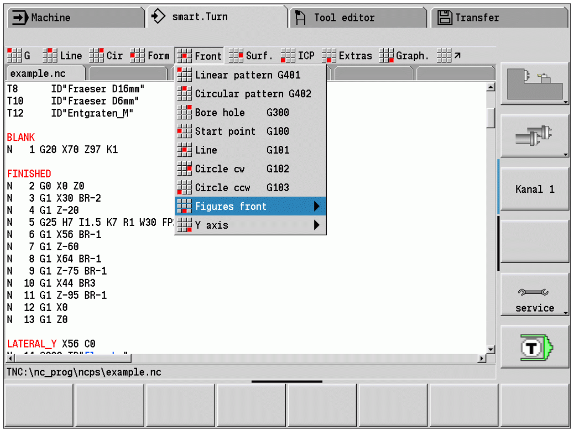
The **Geo(metry) pull-down menus** contain functions for contour description. The pull-down menus are called by pressing the "Geo" menu in DIN/ISO mode.

Overview of the functions:

- **G**: Direct entry of a G function
- **Line**: Direct entry of a line segment G1
- **Circle**: Description of a circular arc (G2, G3, G12, G13)
- **Form**: Description of form elements
- **Front**: Functions for contour descriptions on the front face
- **Surface**: Functions for contour descriptions on the lateral surface
- ICP, Extras, Graph.: See "Shared menu items" on page 40.



▶ Back to the DIN/ISO main menu

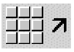


"Machining" pull-down menus

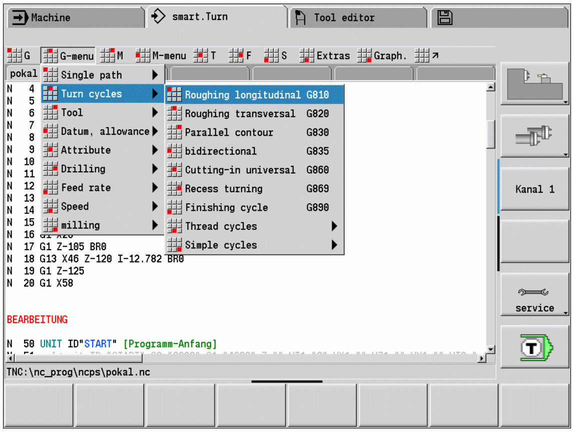
The **"Mach (ining)" pull-down menus** contain functions for programming the machining operation. The pull-down menus are called by pressing the "Mach" menu in DIN/ISO mode.

Overview of the functions:

- **G**: Direct entry of a G function
- **G menu**: Pull-down menus for machining tasks
- **M**: Direct entry of an M function
- **M menu**: Pull-down menus for switching tasks
- **T**: Direct tool call
- **F**: Feed per revolution G95
- **S**: Cutting speed G96
- Extras, Graph.: See "Shared menu items" on page 40.



▶ Back to the DIN/ISO main menu



4.2 Definition of workpiece blank

Chuck part bar/tube G20-Geo

G20 defines the contour of a cylinder/hollow cylinder.

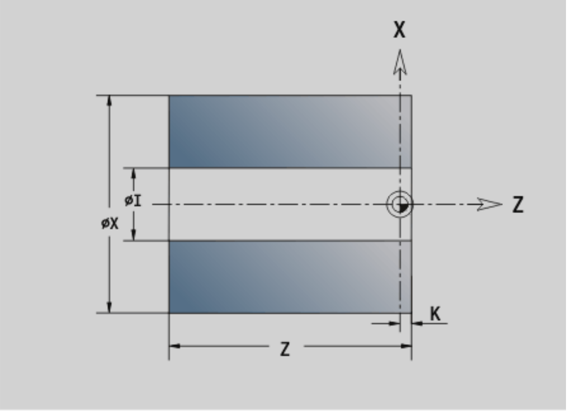
Parameters

X
- Cylinder/hollow cylinder diameter
- Diameter of circumference of a polygonal blank

Z Length of the blank

K Right edge (distance between workpiece zero point and right edge)

I Inside diameter of hollow cylinders



Example: G20-Geo

```
. . .
BLANK
N1 G20 X80 Z100 K2 I30 [hollow cylinder]
. . .
```

Cast part G21-Geo

G21 generates the contour of the blank part from the contour of the finished part—plus the equidistant oversize P.

Parameters

P Equidistant oversize (reference: finished part contour)

Q Bore hole Y/N (default: 0)
- 0: Without hole
- 1: With hole

> G21 cannot be used to describe an "auxiliary blank."

Example: G21-Geo

```
. . .
BLANK
N1 G21 P5 Q1 [cast blank]
. . .
FINISHED PART
N2 G0 X30 Z0
N3 G1 X50 BR-2
N4 G1 Z-40
N5 G1 X65
N6 G1 Z-70
. . .
```

4.3 Basic contour elements

Starting point of turning contour G0-Geo

G0 defines the starting point of a turning contour.

Parameters

X Contour starting point (diameter value)

Z Contour starting point

Example: G0-Geo

```
. . .
FINISHED PART
N2 G0 X30 Z0 [starting point of contour]
N3 G1 X50 BR-2
N4 G1 Z-40
N5 G1 X65
N6 G1 Z-70
. . .
```

Machining attributes for form elements

All the basic contour elements contain the chamfer/rounding form element (BR). You can define machining attributes for this form element and for all the other form elements (recesses, undercuts).

Parameters

BE Special feed factor for the chamfer/rounding arc during the finishing cycle (default: 1)

Special feed rate = active feed rate * BE

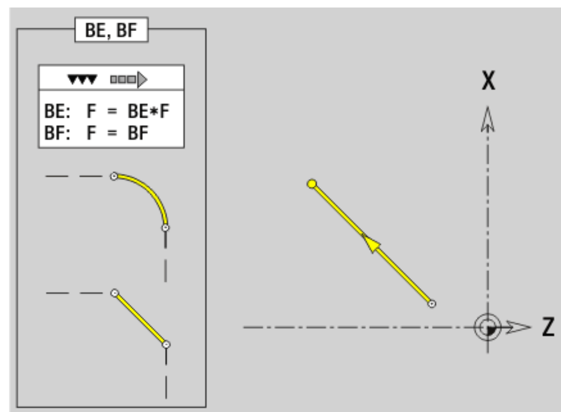
BF Special feed rate for the chamfer/rounding arc during the finishing cycle (default: no special feed rate)

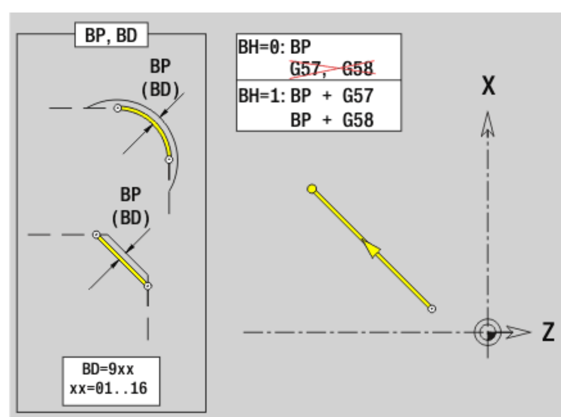
BD Additive compensation number for the chamfer/rounding arc (901-916)

BP Equidistant oversize (at constant distance) for the chamfer/rounding arc

BH Type of oversize for the chamfer/rounding arc

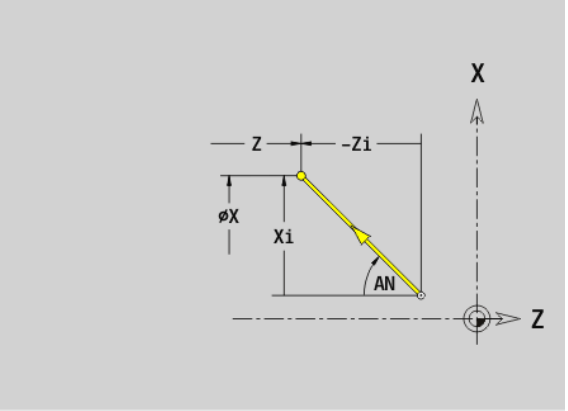
- 0: Absolute oversize
- 1: Additive oversize

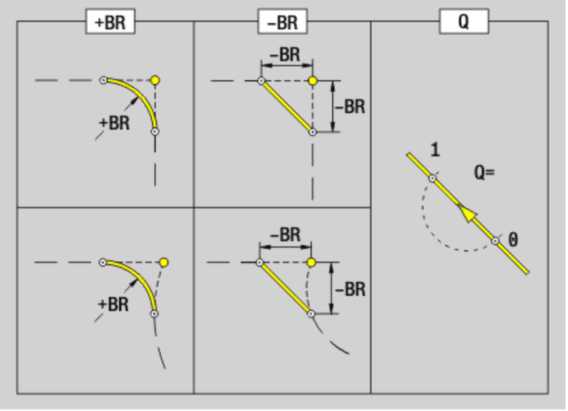




Line segment in a contour G1-Geo

G1 defines a line segment in a turning contour.





Parameters

- X End point of contour element (diameter value)
- Z End point of contour element
- AN Angle to rotary axis (for angle direction see graphic support window)
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
  - 0: Near point of intersection
  - 1: Far point of intersection
- BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
  - No entry: Tangential transition
  - BR=0: No tangential transition
  - BR>0: Rounding radius
  - BR<0: Width of chamfer
- BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)
- FP Do not machine element (only necessary for TURN PLUS):
  - 0: Do not machine basic element (straight line)
  - 1: Do not machine overlay element (e.g. chamfer or rounding)
  - 2: Do not machine basic/overlay element
- IC Measuring cut oversize (measuring cut diameter)
- KC Length of measuring cut
- HC Measuring cut counter: Number of workpieces after which a measurement is performed

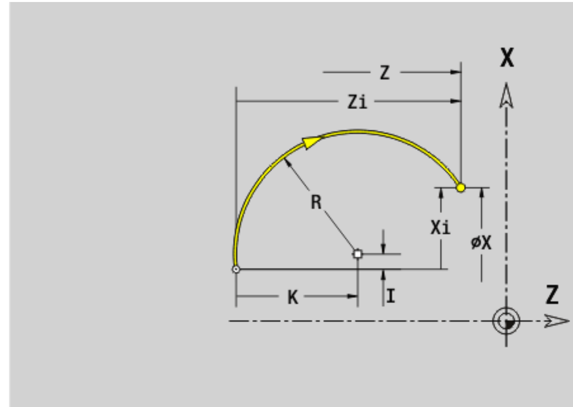
Example: G1-Geo

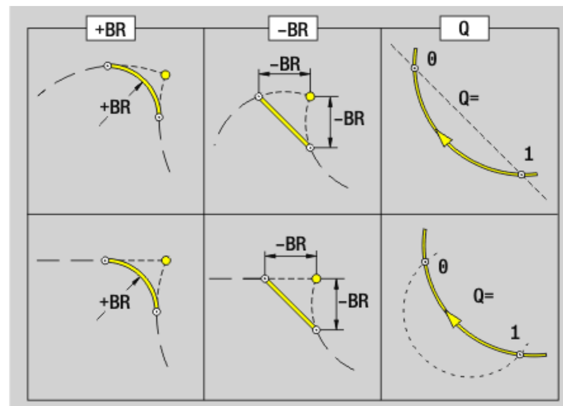
. . .	
FINISHED PART	
N2 G0 X0 Z0	Starting point
N3 G1 X50 BR-2	Perpendicular line with chamfer
N4 G1 Z-20 BR2	Horizontal line with radius
N5 G1 X70 Z-30	Oblique cut with absolute target coordinates
N6 G1 Zi-5	Horizontal line segment, incremental
N7 G1 Xi10 AN30	Incremental and angle
N8 G1 X92 Zi-5	Incremental and absolute mixed
N9 G1 X? Z-80	Calculate the X coordinate
N10 G1 X100 Z-100 AN10	End point and angle with unknown starting point
. . .	

Circular arc of turning contour G2/G3-Geo

G2/G3 defines a circular arc in a contour with **incremental** center dimensioning. Direction of rotation (see help graphic):

- G2: In clockwise direction
- G3: In counterclockwise direction





Parameters

X End point of contour element (diameter value)
Z End point of contour element
I Center (distance from starting point to center as radius)
K Center (distance from starting point to center)
R Radius
Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

FP Do not machine element (only necessary for TURN PLUS):

- 0: Do not machine basic element (circle)
- 1: Do not machine overlay element (e.g. chamfer or rounding)
- 2: Do not machine basic/overlay element

Programming X, Z: Absolute, incremental, modal or "?"

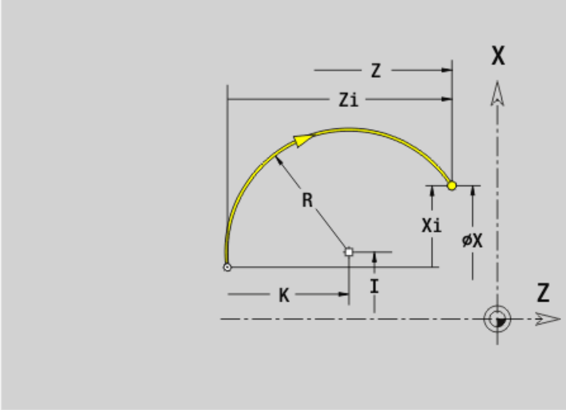
Example: G2-, G3-Geo

. . .	
FINISHED PART	
N1 G0 X0 Z-10	
N2 G3 X30 Z-30 R30	Target point and radius
N3 G2 X50 Z-50 I19.8325 K-2.584	Target point and center, incremental
N4 G3 Xi10 Zi-10 R10	Target point (incremental) and radius
N5 G2 X100 Z? R20	Unknown target point coordinate
N6 G1 Xi-2.5 Zi-15	
. . .	

Circular arc of turning contour G12/G13-Geo

G12/G13 defines a circular arc in a contour with **absolute** center dimensioning. Direction of rotation (see help graphic):

- G12: In clockwise direction
- G13: In counterclockwise direction



Parameters

X End point of contour element (diameter value)
Z End point of contour element
I Center (radius dimension)
K Center
R Radius
Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

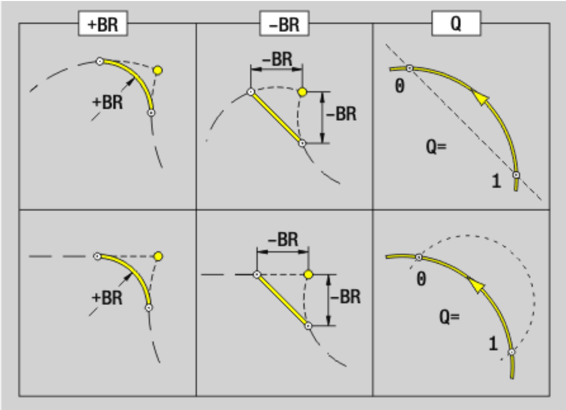
- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)
FP Do not machine element (only necessary for TURN PLUS):

- 0: Do not machine basic element (straight line)
- 1: Do not machine overlay element (e.g. chamfer or rounding)
- 2: Do not machine basic/overlay element



Programming X, Z: Absolute, incremental, modal or "?"

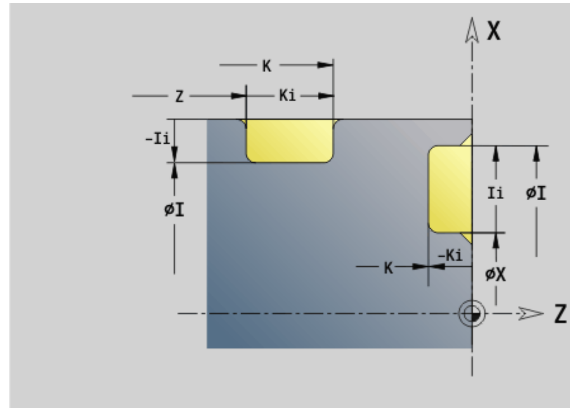
Example: G12-, G13-Geo

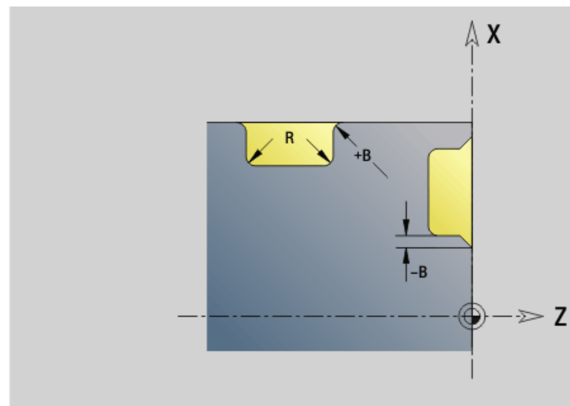
. . .	
FINISHED PART	
N1 G0 X0 Z-10	
. . .	
N7 G13 Xi-15 Zi15 R20	Target point (incremental) and radius
N8 G12 X? Z? R15	Only the radius is known
N9 G13 X25 Z-30 R30 BR10 Q1	Rounding arc in transition and selection of intersections
N10 G13 X5 Z-10 I22.3325 K-12.584	Target point and center, absolute
. . .	

4.4 Contour form elements

Recess (standard) G22-Geo

G22 defines a recess on the previously programmed paraxial reference element.





Parameters

X Starting point of recess on the face (diameter)

Z Starting point of recess on the lateral surface

I Inside corner (diameter value)

- Recessing on face: End point of the recess
- Recess on lateral surface: Recess floor

K Inside corner

- Recess on face: Recess base
- Recessing on lateral surface: End point of the recess

Ii Inside corner—incremental (pay attention to algebraic sign!)

- Recess on face: Recess width
- Recess on lateral surface: Recess depth

Ki Inside corner—incremental (pay attention to algebraic sign!)

- Recess on face: Recess depth
- Recess on lateral surface: Recess width

B Outside radius/chamfer at both sides of the recess (default: 0)

- B>0: Rounding radius
- B<0: Chamfer width

R Inside radius in both corners of recess (default: 0)

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

FP Do not machine element (only necessary for TURN PLUS):

- 1: Do not machine recess

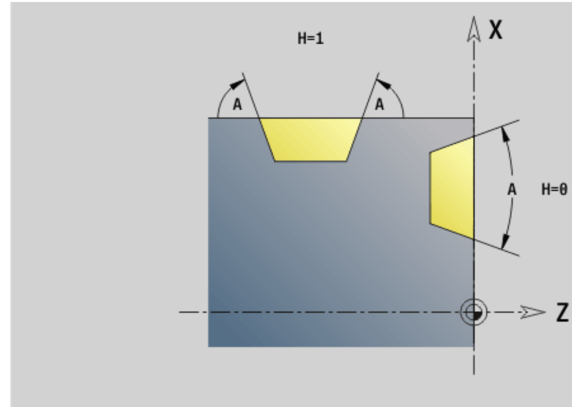
Program only X or Z.

Example: G22-Geo

FINISHED PART	
N1 G0 X40 Z0	
N2 G1 X80	
N3 G22 X60 I70 Ki-5 B-1 R0.2	Recess on face, depth is incremental
N4 G1 Z-80	
N5 G22 Z-20 I70 K-28 B1 R0.2	Longitudinal recess, width is absolute
N6 G22 Z-50 Ii-8 Ki-12 B0.5 R0.3	Longitudinal recess, width is incremental
N7 G1 X40	
N8 G1 Z0	
N9 G22 Z-38 Ii6 K-30 B0.5 R0.2	Longitudinal recess, inside
. . .	

Recess (general) G23-Geo

G23 defines a recess on the previously programmed linear reference element. The reference element can also be oblique.



Parameters

H Type of recess (default: 0)
- 0: Symmetrical recess
- 1: Relief turn

X Center point of recess on the face (diameter)

No input: Position is calculated

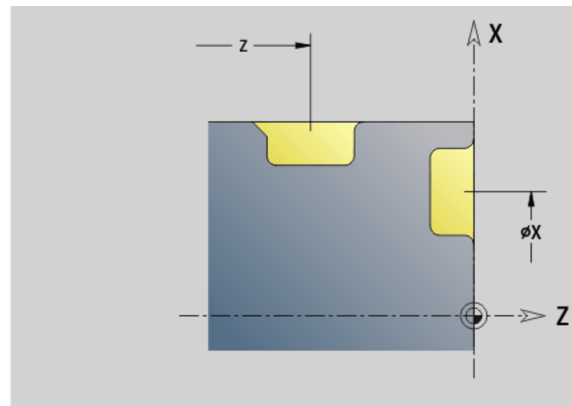
Z Center point of recess on the lateral surface

No input: Position is calculated

I Recess depth and recess position
- I>0: Recess at the right of the reference element
- I<0: Recess at the left of the reference element

K Recess width (without chamfer/rounding arc)

U Recess diameter (diameter of recess base). Use U only if the reference element runs parallel to the Z axis



A Recess angle (default: 0)
- H=0: Angle between recess edges (0° <= A < 180°)
- H=1: Angle between reference line and recess edge (0° < A <= 90°)

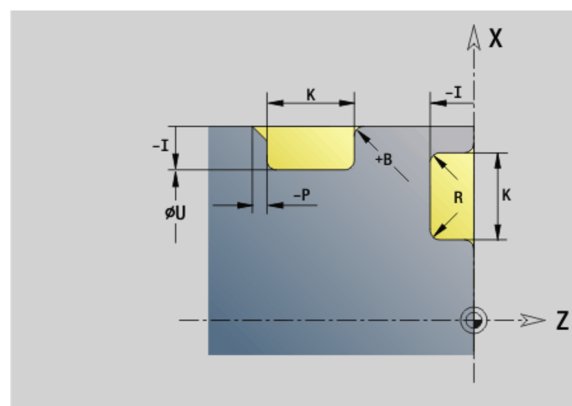
B Outside radius/chamfer at corner near the starting point (default: 0)
- B>0: Rounding radius
- B<0: Chamfer width

P Outside radius/chamfer at corner far from the starting point (default: 0)
- P>0: Radius of the rounding arc
- P<0: Chamfer width

R Inside radius in both corners of recess (default: 0)

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

FP Do not machine element (only necessary for TURN PLUS):
- 1: Do not machine recess



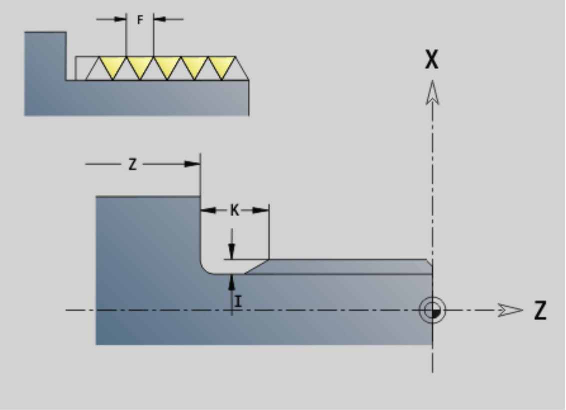
The Control refers the recess depth to the reference element. The recess base runs parallel to the reference element.

Example: G23-Geo

. . .	
FINISHED PART	
N1 G0 X40 Z0	
N2 G1 X80	
N3 G23 H0 X60 I-5 K10 A20 B-1 P1 R0.2	Recess on face, depth is incremental
N4 G1 Z-40	
N5 G23 H1 Z-15 K12 U70 A60 B1 P-1 R0.2	Longitudinal recess, width is absolute
N6 G1 Z-80 A45	
N7 G23 H1 X120 Z-60 I-5 K16 A45 B1 P-2 R0.4	Longitudinal recess, width is incremental
N8 G1 X40	
N9 G1 Z0	
N10 G23 H0 Z-38 I-6 K12 A37.5 B-0.5 R0.2	Longitudinal recess, inside
. . .	

Thread with undercut G24-Geo

G24 defines a linear basic element with a longitudinal thread and subsequent thread undercut (DIN 76). The thread is an outside or inside thread (metric ISO fine-pitch thread DIN 13 Part 2, Series 1).



Parameters

- F Thread pitch
- I Undercut depth (radius)
- K Width of undercut
- Z End point of the undercut

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

FP Do not machine element (only necessary for TURN PLUS):

- 1: Do not machine element

- Program G24 only in closed contours.
- The thread is machined with G31.

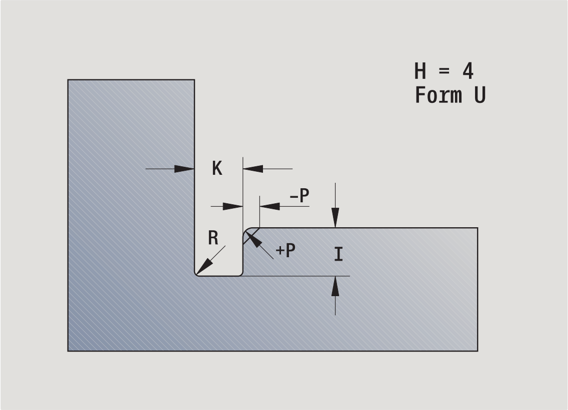
Example: G24-Geo

. . .	
FINISHED PART	
N1 G0 X40 Z0	
N2 G1 X40 BR-1.5	Starting point for thread
N3 G24 F2 I1.5 K6 Z-30	Thread with undercut
N4 G1 X50	Next transverse element
N5 G1 Z-40	
. . .	

Undercut contour G25-Geo

G25 generates the undercut contours listed below. The undercuts are only possible in inside contour corners in which the transverse element is parallel to the X axis. Program G25 after the first element. You specify the undercut type in parameter H.

Undercut type U (H=4)



Parameters

H Undercut type U: H=4

I Undercut depth (radius)

K Width of undercut

R Inside radius in both corners of recess (default: 0)

P Outside radius/chamfer (default: 0)

- P>0: Radius of the rounding arc
- P<0: Chamfer width

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

FP Do not machine element (only necessary for TURN PLUS):

- 1: Do not machine undercut

Example: Call G25-Geo type U

```
. . .
N.. G1 Z-15 [longitudinal element]
N.. G25 H4 I2 K4 R0.4 P-0.5 [type U]
N.. G1 X20 [transverse element]
. . .
```

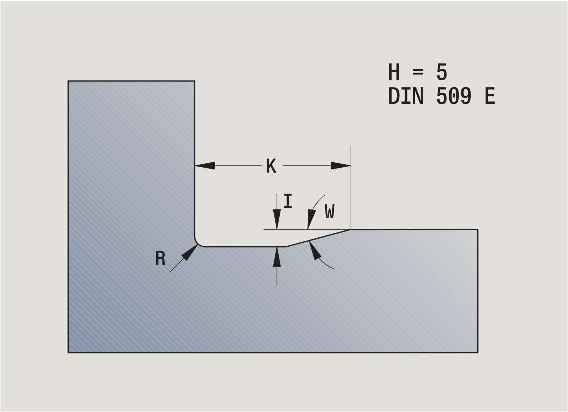
Undercut DIN 509 E (H=0.5)

Parameters

- H Undercut type DIN 509 E: H=0 or H=5
- I Undercut depth (radius)
- K Width of undercut
- R Undercut radius (in both corners of the undercut)
- W Undercut angle

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

The Control uses the diameter to calculate the parameters that you do not define.



Example: Call G25-Geo DIN 509 E

```
. . .
N.. G1 Z-15 [longitudinal element]
N.. G25 H5 [DIN 509 E]
N.. G1 X20 [transverse element]
. . .
```

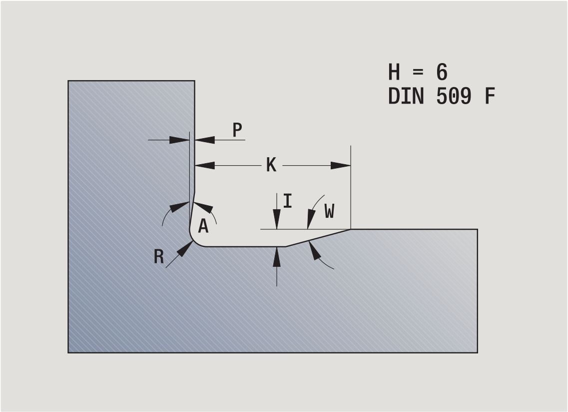
Undercut DIN 509 F (H=6)

Parameters

- H Undercut type DIN 509 F: H=6
- I Undercut depth (radius)
- K Width of undercut
- R Undercut radius (in both corners of the undercut)
- P Face depth
- W Undercut angle
- A Transverse angle

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)

The Control uses the diameter to calculate the parameters that you do not define.



Example: Call G25-Geo DIN 509 F

```
. . .
N.. G1 Z-15 [longitudinal element]
N.. G25 H6 [DIN 509 F]
N.. G1 X20 [transverse element]
. . .
```

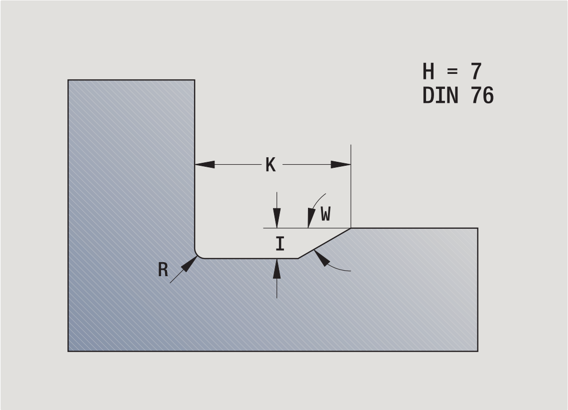
Undercut DIN 76 (H=7)

Program only FP. All the other values are automatically calculated from the thread pitch in the standard table if they are not defined.

Parameters

- H Undercut type DIN 76: H=7
- I Undercut depth (radius)
- K Width of undercut
- R Undercut radius in both corners of the undercut (default: R=0.6*I)
- W Undercut angle (default: 30°)
- FP Thread pitch

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)



Example: Call G25-Geo DIN 76

```
. . .
N.. G1 Z-15 [longitudinal element]
N.. G25 H7 FP2 [DIN 76]
N.. G1 X20 [transverse element]
. . .
```

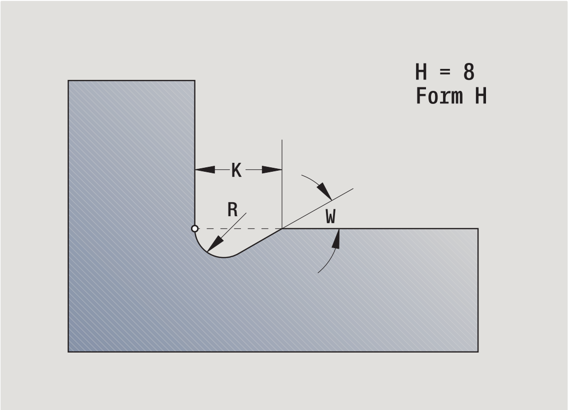
Undercut type H (H=8)

If you do not enter W, the angle will be calculated on the basis of K and R. The end point of the undercut is then located at the "contour corner."

Parameters

- H Undercut type H: H=8
- K Width of undercut
- R Undercut radius—no value: The circular element is not machined
- W Plunge angle—no value: W is calculated

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)



Example: Call G25-Geo type H

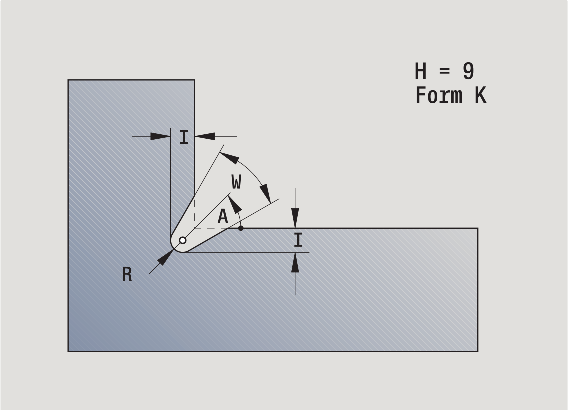
```
. . .
N.. G1 Z-15 [longitudinal element]
N.. G25 H8 K4 R1 W30 [type H]
N.. G1 X20 [transverse element]
. . .
```

Undercut type K (H=9)

Parameters

H Undercut type K: H=9
I Undercut depth
R Undercut radius—no value: The circular element is not machined
W Undercut angle
A Angle to longitudinal axis (default: 45°)

BE, BF, BD, BP and BH (see "Machining attributes for form elements" on page 195)



Example: Call G25-Geo type K

```
. . .
N.. G1 Z-15 [longitudinal element]
N.. G25 H9 I1 R0.8 W40 [type K]
N.. G1 X20 [transverse element]
. . .
```

Thread (standard) G34-Geo

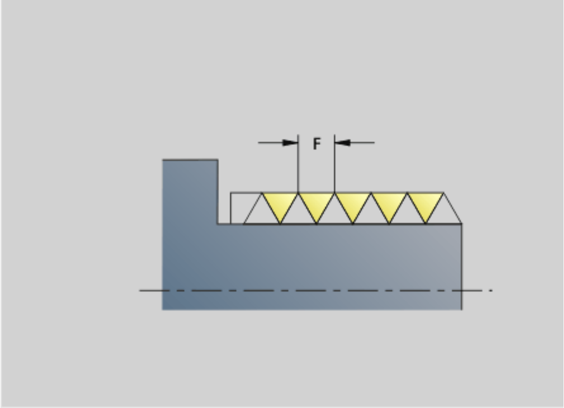
G34 defines a simple or an interlinked external or internal thread (metric ISO fine-pitch thread DIN 13 Series 1). The Control calculates all the required values.

Parameters

F   Thread pitch (default: pitch from the standard table)

Threads are concatenated by programming several G1/G34 blocks after each other.

- You need to program a linear contour element as a reference before G34 or in the NC block containing G34.
- Machine the thread with G31.

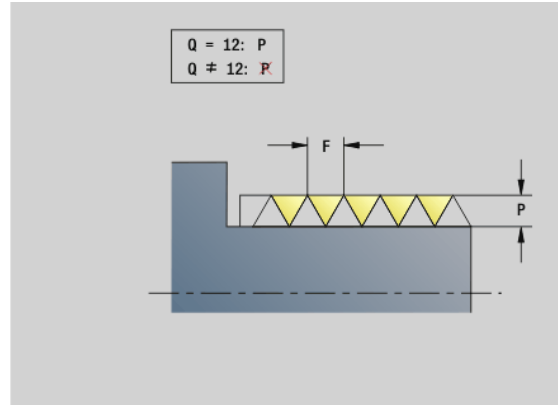


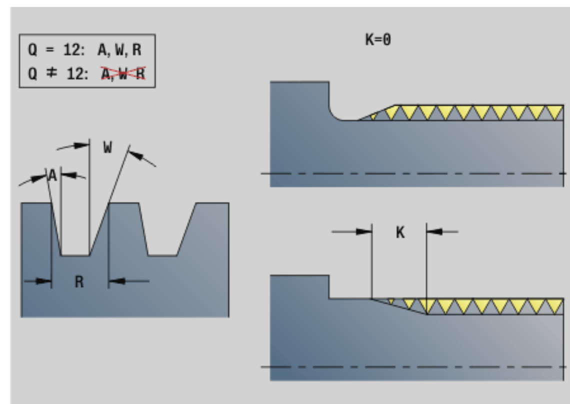
Example: G34

```
. . .
FINISHED PART
N1 G0 X0 Z0
N2 G1 X20 BR-2
N3 G1 Z-30
N4 G34 [metric ISO]
N5 G25 H7 I1.7 K7
N6 G1 X30 BR-1.5
N7 G1 Z-40
N8 G34 F1.5 [metric ISO fine-pitch thread]
N9 G25 H7 I1.5 K4
N10 G1 X40
N11 G1 Z-60
. . .
```

Thread (general) G37-Geo

G37 defines the different types of thread. Multi-start threads and concatenated threads are possible. Threads are concatenated by programming several G01/G34 blocks after each other.





Parameters

Q Type of thread (default: 1)

- 1: Metric ISO fine-pitch thread (DIN 13 Part 2, Series 1)
- 2. Metric ISO thread (DIN 13 Part 1, Series 1)
- 3: Metric ISO tapered thread (DIN 158)
- 4: Metric ISO tapered fine-pitch thread (DIN 158)
- 5: Metric ISO trapezoid thread (DIN 103 Part 2, Series 1)
- 6: Flat metric trapezoid thread (DIN 380 Part 2, Series 1)
- 7: Metric buttress thread (DIN 513 Part 2, Series 1)
- 8: Cylindrical round thread (DIN 405 Part 1, Series 1)
- 9: Cylindrical Whitworth thread (DIN 11)
- 10: Tapered Whitworth thread (DIN 2999)
- 11: Whitworth pipe thread (DIN 259)
- 12: Nonstandard thread
- 13: UNC US coarse thread
- 14: UNF US fine-pitch thread
- 15: UNEF US extra-fine-pitch thread
- 16: NPT US taper pipe thread
- 17: NPTF US taper dryseal pipe thread
- 18: NPSC US cylindrical pipe thread with lubricant
- 19: NPFS US cylindrical pipe thread without lubricant

F Thread pitch

- Required for Q=1, 3 to 7, 12.
- For other thread types, F is calculated from the diameter if it was not programmed.

P Thread depth—enter only for Q=12

K Run-out length for threads without undercut (default: 0)

D Reference point (default: 0)

- 0: Runout of thread at the end of the reference element
- 1: Runout of thread at the beginning of the reference element

H Number of thread turns (default: 1)

A Thread angle at left—enter only for Q=12

W Thread angle at right—enter only for Q=12

R Thread width—enter only for Q=12

E Variable pitch (default: 0)

Increase/decrease the pitch per revolution by E.

V Direction of thread

- 0: Right-hand thread
- 1: Left-hand thread

Example: G37

```
. . .
FINISHED PART
N1 G0 X0 Z0
N2 G1 X20 BR-2
N3 G1 Z-30
N4 G37 Q2[metric ISO]
N5 G25 H7 I1.7 K7
N6 G1 X30 BR-1.5
N7 G1 Z-40
N8 G37 F1.5 [metric ISO fine-pitch thread]
N9 G25 H7 FP1.5
N10 G1 X40
N11 G1 Z-60
. . .
```

- Before G37, program a linear contour element as a reference.
- Machine the thread with G31.
- For standard threads, the parameters P, R, A and W are defined by the Control.
- Use Q=12 if you wish to use individual parameters.

Danger of collision!
The thread is generated to the length of the reference element. Another linear element without undercut is to be programmed as overrun.

Example: G37 Concatenated

```
. . .
AUXILIARY CONTOUR ID"G37_Concatenated"
N37 G0 X0 Z0
N  38 G1 X20
N  39 G1 Z-30
N  40 G37 F2[metric ISO]
N  41 G1 X30 Z-40
N  42 G37 Q2
N  43 G1 Z-70
N  44 G37 F2
. . .
```

Bore hole (centric) G49-Geo

G49 defines a single hole with countersink and thread **at the turning center** (front or rear face). The G49 hole is a form element, not part of the contour.

Parameters

Z Starting position for hole (reference point)
B Hole diameter
P Depth of hole (excluding point)
W Point angle (default: 180°)
R Sinking diameter
U Sinking depth
E Sinking angle
I Thread diameter
J Thread depth
K Thread chamfer
F Thread pitch
V Left-hand or right-hand thread (default: 0)

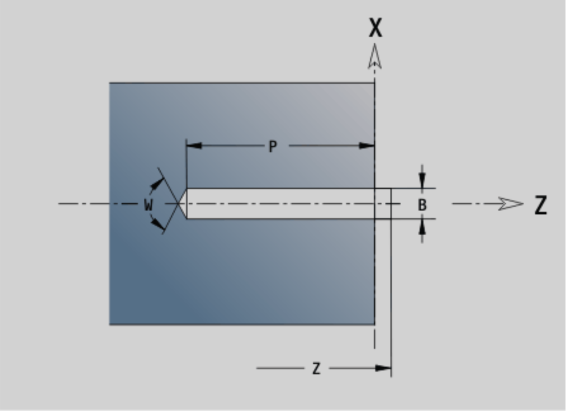
- 0: Right-hand thread
- 1: Left-hand thread

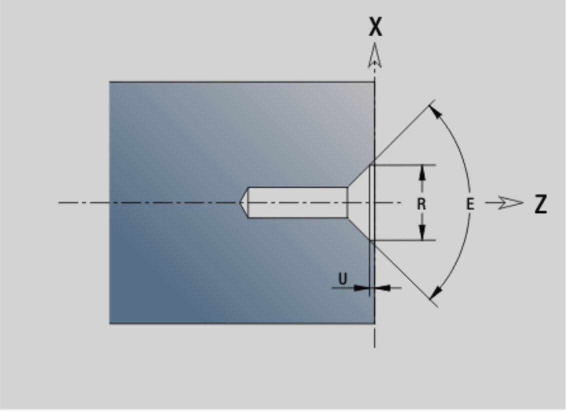
A Angle corresponding to the position of the hole (default: 0)

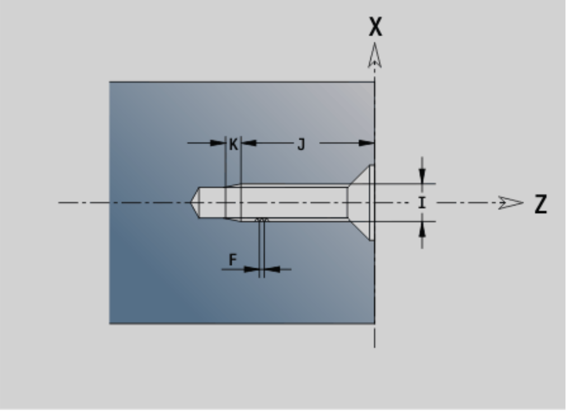
- A=0°: Front face
- A=180°: Rear side

O Centering diameter

- Program G49 in the **FINISHED PART** section, not in **AUXILIARY CONTOUR**, **FRONT** or **REAR SIDE**.
- Machine the G49 hole with G71...G74.







4.5 Attributes for contour description

Overview of attributes for contour description		
G38	Special feed factor for basic elements and form elements—modal	Page 213
G52	Equidistant oversize for basic elements and form elements—modal	Page 215
G95	Finishing feed rate for basic elements and form elements—modal	Page 216
G149	Additive compensation for basic elements and form elements—modal	Page 216

- Once programmed, G38-, G52-, G95- and G149-Geo remain in effect for all contour elements until the function is programmed again without defining parameters.
- For form elements, you can program different attributes directly in the definition of the form element (see "Machining attributes for form elements" on page 195).
- The attributes for contour description influence the finishing feed rate of the Cycles G869 and G890, not the finishing feed rate in recessing cycles.

Feed rate reduction factor G38-Geo

G38 activates the special feed rate for the finishing cycle G890. The special feed rate applies to basic contour elements and form elements. It is a modal function.

Parameter

E Special feed factor (default: 1)

Special feed rate = active feed rate * E

- G38 is a modal function.
- Program G38 **before** the contour element for which it is intended.
- G38 **replaces** a special feed rate.
- To cancel the special feed factor, program G38 without parameters.

Attributes for superimposed elements G39-Geo

G39 influences the finishing feed rate of G890 with the form elements:

- Chamfers/rounding arcs (for connecting basic elements)
- Undercuts
- Recesses

Affected machining: Special feed rate, surface roughness, additive D compensation, equidistant oversizes.

Parameters

F Feed per revolution

V Type of surface roughness (see also DIN 4768)

- 1: General surface roughness (profile depth) Rt1
- 2: Surface roughness Ra
- 3: Surface roughness Rz

RH Surface roughness [µm, inch mode: µinch]

D Number of the additive compensation (901 <= D <= 916)

P Oversize (radius)

H P applies as an absolute or additive value (default: 0)

- 0: P replaces G57/G58 oversizes
- 1: P is added to G57/G58 oversizes

E Special feed factor (default: 1)

Special feed rate = active feed rate * E

- Use surface roughness (V, RH), finishing feed rate (F) and special feed rate ("E") alternately!
- G39 is a non-modal function.
- Program G39 **before** the contour element for which it is intended.
- G50 preceding a cycle (MACHINING section) cancels a finishing oversize programmed for that cycle with G39.

Function G39 can be replaced by directly entering the attributes in the contour elements dialog. The function is necessary to execute imported programs correctly.

Separation point G44

During automatic program creation with TURN PLUS, you can define the separation point for rechucking with function G44.

Parameter
D Location of separation point:

- 0: Start of the basic element as separation point
- 1: Target of the basic element as separation point

If no separation point was defined, TURNplus uses the largest diameter as separation point for outside machining and the smallest diameter as separation point for inside machining.

Oversize G52-Geo

G52 defines an equidistant oversize that applies to basic contour elements and form elements and is taken into consideration in G810, G820, G830, G860 and G890.

Parameters
P Oversize (radius)
H P applies as an absolute or additive value (default: 0)

- 0: P replaces G57/G58 oversizes
- 1: P is added to G57/G58 oversizes

- G52 is a modal function.
- Program G52 **in** the NC block for which it is intended.
- G50 preceding a cycle (**MACHINING** section) cancels an oversize programmed for that cycle with G52.

Feed per revolution G95-Geo

G95 influences the finishing feed rate of G890 for basic contour elements and form elements.

Parameter
F Feed per revolution

- The G95 finishing feed rate replaces a finishing feed rate defined in the machining program.
- G95 is a modal function.
- To cancel a finishing feed rate set with G95, program G95 without an input value.

Example: Attributes in contour description G95

```
. . .
FINISHED PART
N1 G0 X0 Z0
N2 G1 X20 BR-1
N3 G1 Z-20
N4 G25 H5 I0.3 K2.5 R0.6 W15
N5 G1 X40 BR-1
N6 G95 F0.08
N7 G1 Z-40
N8 G25 H5 I0.3 K2.5 R0.6 W15 BF0
N9 G95
N10 G1 X58 BR-1
N11 G1 Z-60
. . .
```

Additive compensation G149-Geo

G149 followed by a D number activates/deactivates an additive compensation function. The Control manages the 16 tool-independent compensation values in an internal table. The compensation values are managed in the Program Run mode (see "Program Run mode" in the User's Manual).

Parameter
D Additive compensation (default: D900)

- D=900: Deactivates the additive compensation
- D=901 to 916: Activates the additive compensation D

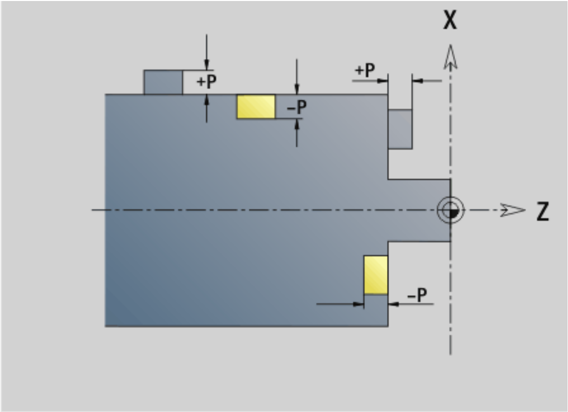
- Note the direction of contour description.
- Additive compensation is effective from the block in which G149 is programmed.
- Additive compensation remains in effect up to:
  - the next G149 D900,
  - up to the end of the finished part description.

Example: Attributes in contour description G149

```
. . .
FINISHED PART
N1 G0 X0 Z0
N2 G1 X20 BR-1
N3 G1 Z-20
N4 G25 H5 I0.3 K2.5 R0.6 W15
N5 G1 X40 BR-1
N6 G149 D901
N7 G1 Z-40
N8 G25 H5 I0.3 K2.5 R0.6 W15 BD900
N9 G149 D900
N10 G1 X58 BR-1
N  12 G1 Z-60
. . .
```

4.6 C-axis contours—Fundamentals

Milling contour position



Define the reference plane or the reference diameter in the section code. Specify the depth and position of a milling contour (pocket, island) in the contour definition:

- With **depth P** programmed in the previous G308 cycle.
- Alternatively on figures: Cycle parameter **depth P**.

The **algebraic sign of "P"** defines the position of the milling contour:

- P<0: Pocket
- P>0: Island

Position of milling contour			
Section	**P**	**Surface**	**Milling floor**
FRONT	P<0	Z	Z+P
	P>0	Z+P	Z
REAR SIDE	P<0	Z	Z–P
	P>0	Z–P	Z
SURFACE	P<0	X	X+(P*2)
	P>0	X+(P*2)	X

- X: Reference diameter from the section code
- Z: Reference plane from the section code
- P: Depth from G308 or from cycle parameter

The area milling cycles mill the surface specified in the contour definition. **Islands** within this surface are not taken into consideration.

Contours in more than one plane (hierarchically nested contours):

- A plane begins with G308 and ends with G309.
- G308 defines a new reference plane/reference diameter. The first G308 uses the reference plane defined in the section code. Each following G308 defines a new plane. Calculation:
  New reference plane = Reference plane + P (from previous G308).
- G309 switches back to the previous reference plane.

Start pocket/island G308-Geo

G308 defines a new reference plane / reference diameter in hierarchically nested contours.

Parameters

P Depth for pocket, height for islands

ID Name of the contour for reference from units or cycles

HC Milling/drilling attribute:

- 1: Contour milling
- 2: Pocket milling
- 3: Area milling
- 4: Deburring
- 5: Engraving
- 6: Contour milling and deburring
- 7: Pocket milling and deburring
- 14: Do not machine

Q Milling location:

- 0: On the contour
- 1: Inside/left
- 2: Outside/right

H Direction:

- 0: Up-cut milling
- 1: Climb milling

D Cutter diameter

I Limit diameter

W Angle of the chamfer

BR Chamfer width

RB Return plane

End of pocket/island G309-Geo

G309 defines the end of a reference plane. Every reference plane defined with G308 **must** be ended with G309 (See "Milling contour position" on page 217.).

Example of G308/G309

. . .	
FINISHED PART	
. . .	
FRONT Z0	Define reference plane
N7 G308 P-5 ID"Rectangle"	Beginning of rectangle with depth of –5
N8 G305 XK-5 YK-10 K50 B30 R3 A0	Rectangle
N9 G308 P-10 ID"Circle"	Beginning of "full circle in rectangle" with depth –10
N10 G304 XK-3 YK-5 R8	Full circle
N11 G309	End of full circle
N12 G309	End of rectangle
SURFACE X100	Define reference diameter
N13 G311 Z-10 C45 A0 K18 B8 P-5	Linear slot with depth –5
. . .	

Circular pattern with circular slots

For circular slots in circular patterns you program the pattern positions, the center of curvature, the curvature radius and the position of the slots.

The Control positions the slots as follows:

- Slots are arranged at the distance of the **pattern radius** about the **pattern center** if
  - Pattern center = center of curvature **and**
  - Pattern radius = curvature radius
- Slots are arranged at the distance of the **pattern radius + curvature radius** about the **pattern center** if
  - Pattern center <> center of curvature **or**
  - Pattern radius <> curvature radius

In addition, the position influences the arrangement of the slots:

- **Normal position:** The starting angle of the slot applies as a **relative value** to the pattern position. The starting angle is added to the pattern position.
- **Original position:** The starting angle of the slot applies as an **absolute value** to the pattern position.

The following examples show the programming of a circular pattern with circular slots:

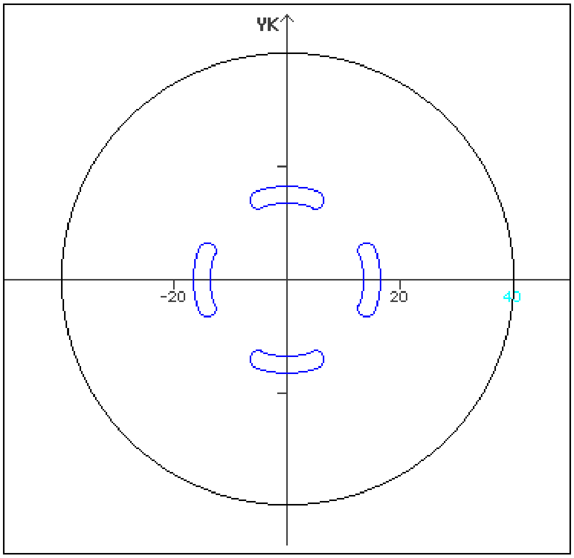
Slot centerline as reference and normal position

Programming:

- Pattern center = center of curvature
- Pattern radius = curvature radius
- Normal position

These commands arrange the slots at the distance of the pattern radius about the pattern center.

Example: Slot centerline as reference, normal position



N.. G402 Q4 K30 A0 XK0 YK0 H0	Circular pattern, normal position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

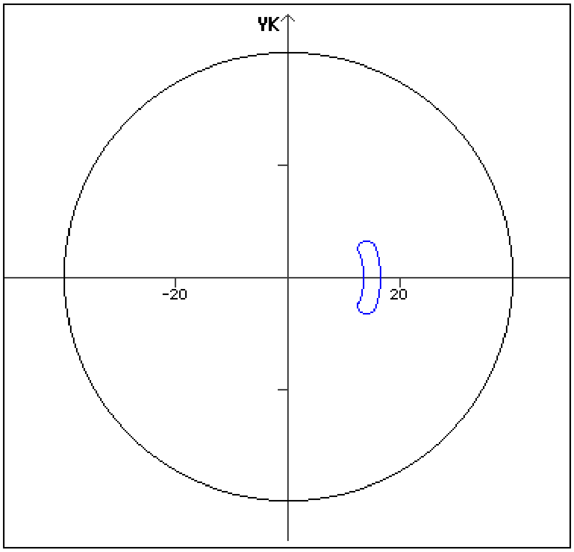
Slot centerline as reference and original position

Programming:

- Pattern center = center of curvature
- Pattern radius = curvature radius
- Original position

These commands arrange all slot at the same position.

Example: Slot centerline as reference, original position



N.. G402 Q4 K30 A0 XK0 YK0 H1	Circular pattern, original position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

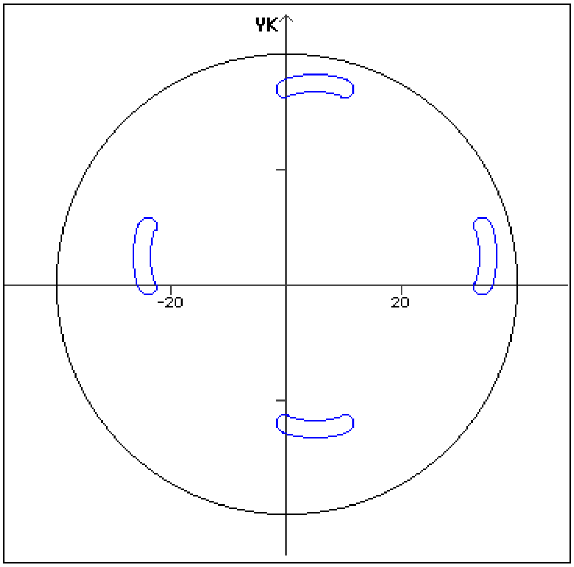
Center of curvature as reference and normal position

Programming:

- Pattern center <> center of curvature
- Pattern radius = curvature radius
- Normal position

These commands arrange the slots at the distance of the pattern radius plus curvature radius about the pattern center.

Example: Center of curvature as reference, normal position



N.. G402 Q4 K30 A0 XK5 YK5 H0	Circular pattern, normal position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

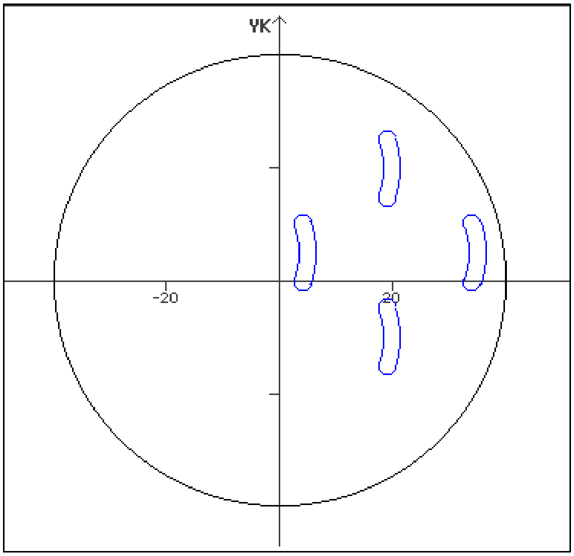
Center of curvature as reference and original position

Programming:

- Pattern center <> center of curvature
- Pattern radius = curvature radius
- Original position

These commands arrange the slots at the distance of the pattern radius plus curvature radius about the pattern center while keeping the starting and ending angle.

Example: Center of curvature as reference and original position



N.. G402 Q4 K30 A0 XK5 YK5 H1	Circular pattern, original position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

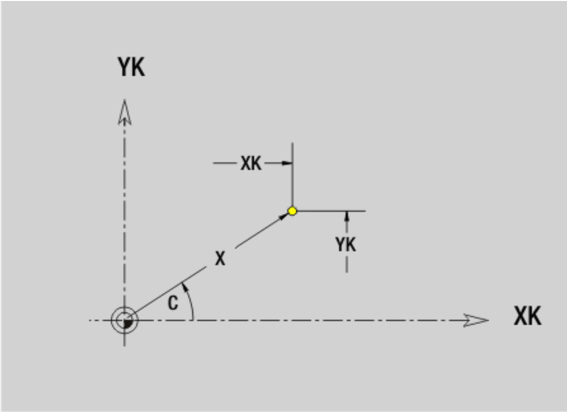
4.7 Front and rear face contours

Starting point of front/rear face contour G100-Geo

G100 defines the starting point of a front or rear face contour.

Parameters

X Starting point in polar coordinates (diameter)
C Starting point in polar coordinates (angular dimension)
XK Starting point in Cartesian coordinates
YK Starting point in Cartesian coordinates



Line segment in front/rear face contour G101-Geo

G101 defines a line segment in a contour on the front face/rear face.

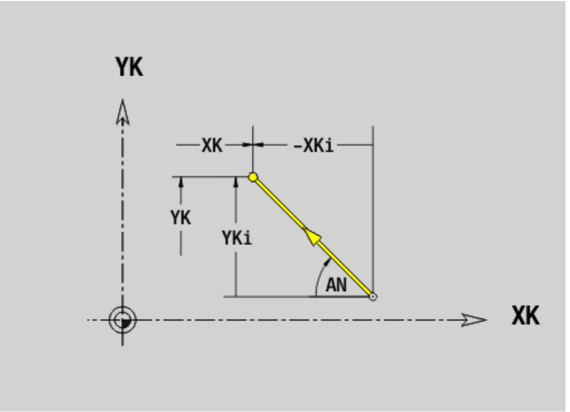
Parameters

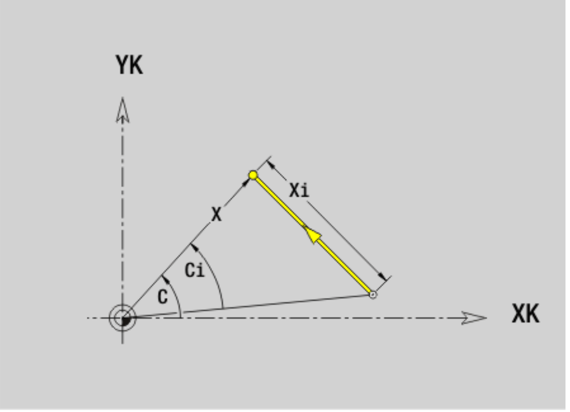
X End point in polar coordinates (diameter)
C End point in polar coordinates (angular dimension)
XK End point in Cartesian coordinates
YK End point in Cartesian coordinates
AN Angle to positive XK axis
BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- 0: Near point of intersection
- 1: Far point of intersection





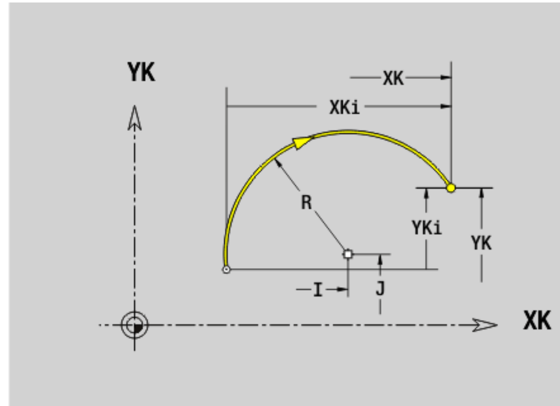
Programming

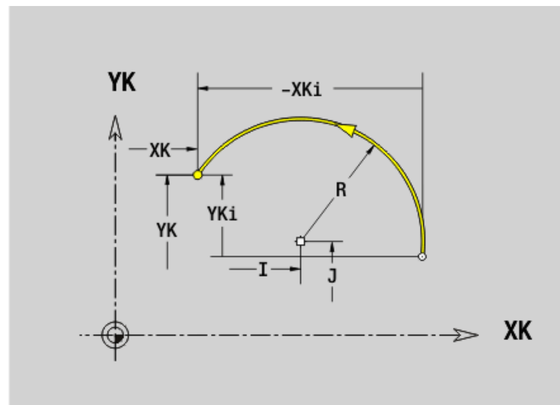
- **X, XK, YX:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal

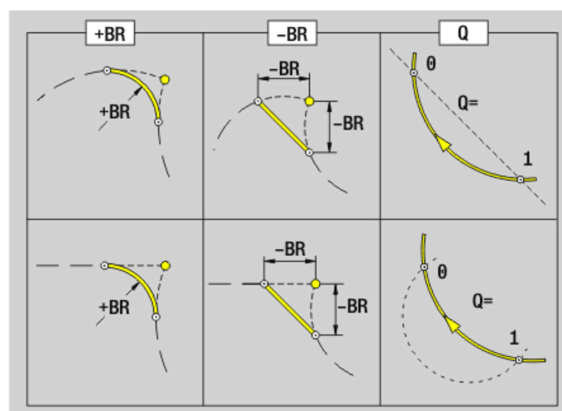
Circular arc in front/rear face contour G102/ G103-Geo

G102/G103 defines a circular arc in a front or rear face contour. Direction of rotation (see help graphic):

- G102: In clockwise direction
- G102: In counterclockwise direction







Parameters

X End point in polar coordinates (diameter)
C End point in polar coordinates (angular dimension)
XK End point in Cartesian coordinates
YK End point in Cartesian coordinates
R Radius
I Center in Cartesian coordinates
J Center in Cartesian coordinates
BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

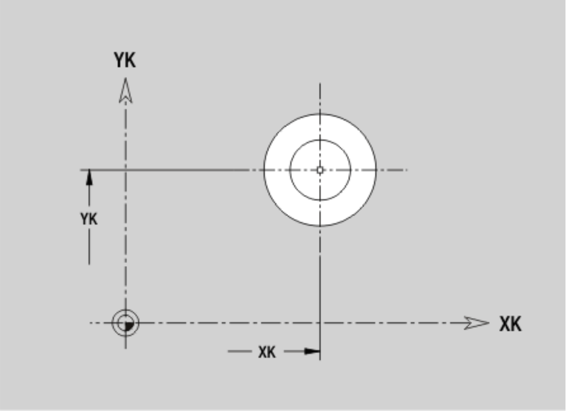
- 0: Near point of intersection
- 1: Far point of intersection

Programming

- **X, XK, YX:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- **I, J:** Absolute or incremental
- End point must not be the starting point (no full circle).

Bore hole on front/rear face G300-Geo

G300 defines a hole with countersinking and thread in a front or rear face contour.



Parameters

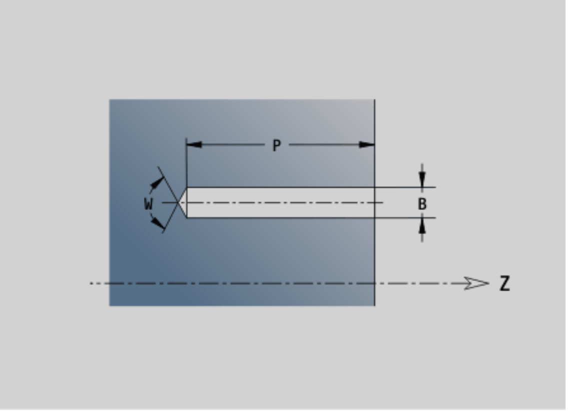
XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
B Hole diameter
P Depth of hole (excluding point)
W Point angle (default: 180°)
R Sinking diameter
U Sinking depth
E Sinking angle
I Thread diameter
J Thread depth
K Thread runout length
F Thread pitch
V Left-hand or right-hand thread (default: 0)

- 0: Right-hand thread
- 1: Left-hand thread

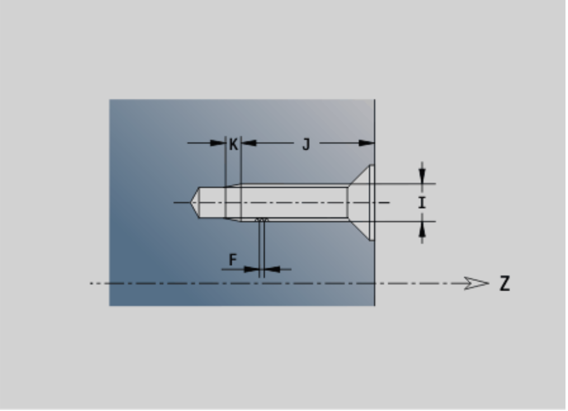
A Angle to Z axis; angle of the hole

- Range for front face: –90° < A < 90° (default: 0°)
- Range for rear side: 90° < A < 270° (default: 180°)

O Centering diameter

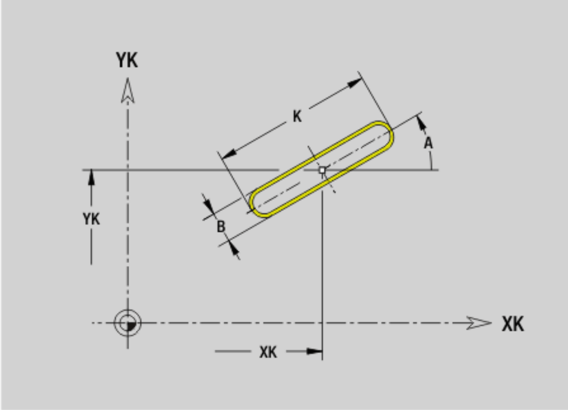


> Machine the G300 holes with G71...G74.



Linear slot on front/rear face G301-Geo

G301 defines a linear slot in a contour on the front or rear face.

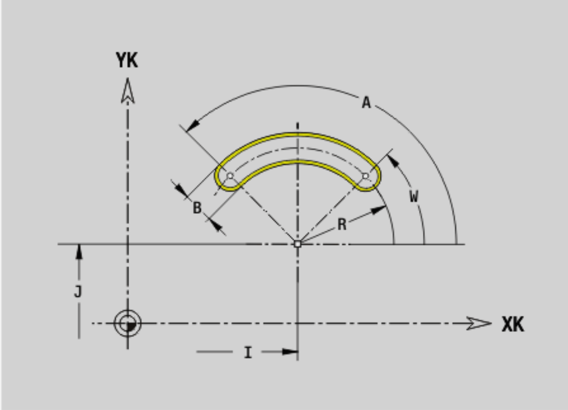


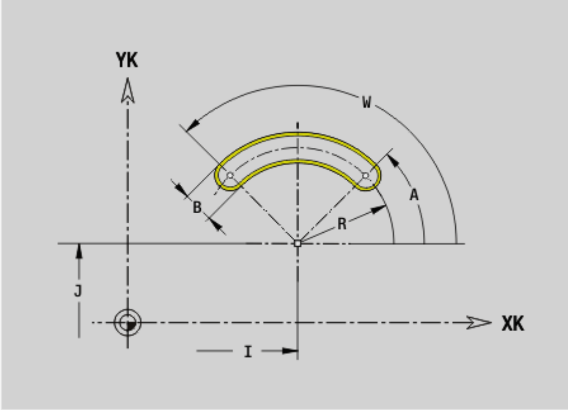
Parameters
- XK Center in Cartesian coordinates
- YK Center in Cartesian coordinates
- X Diameter (center point in polar coordinates)
- C Angle (center point in polar coordinates)
- A Angle to XK axis (default: 0°)
- K Slot length
- B Slot width
- P Depth/height (default: "P" from G308)
  - P<0: Pocket
  - P>0: Island

Circular slot on front/rear face G302/G303-Geo

G302/G303 defines a circular slot in a contour on the front face/rear face.

- G302: Circular slot clockwise
- G303: Circular slot counterclockwise

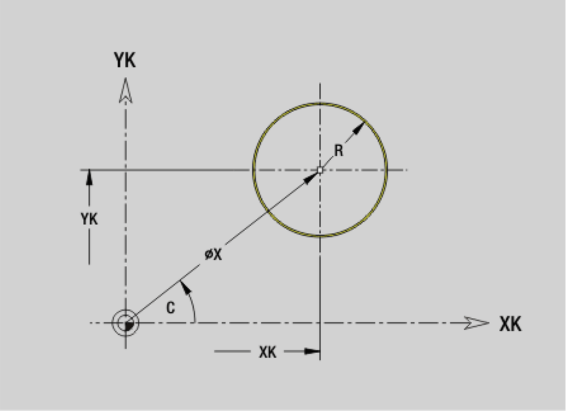




Parameters
- I Center of curvature in Cartesian coordinates
- J Center of curvature in Cartesian coordinates
- X Diameter (center point in polar coordinates)
- C Angle (center point in polar coordinates)
- R Curvature radius (reference: center point path of the slot)
- A Starting angle; reference: XK axis (default: 0°)
- W End angle; reference: XK axis (default: 0°)
- B Slot width
- P Depth/height (default: "P" from G308)
  - P<0: Pocket
  - P>0: Island

Full circle on front/rear face G304-Geo

G304 defines a full circle in a contour on the front face/rear face.



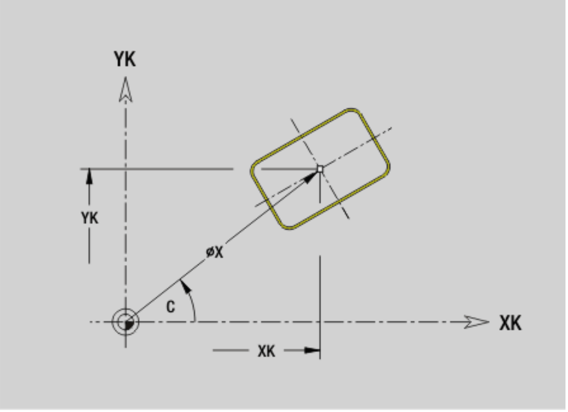
Parameters

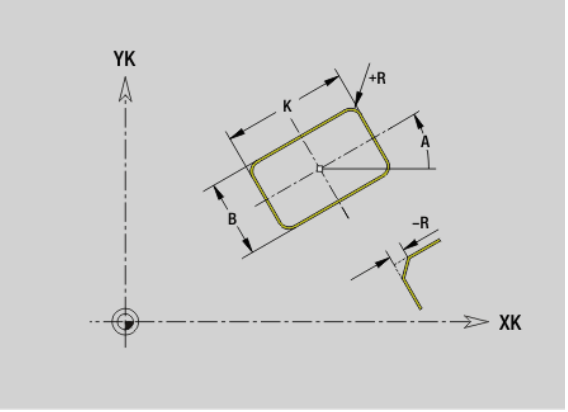
XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
R Radius
P Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island

Rectangle on front/rear face G305-Geo

G305 defines a rectangle in a contour on the front face/rear face.





Parameters

XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
A Angle to XK axis (default: 0°)
K Length
B (Height) width
R Chamfer/rounding arc (default: 0°)

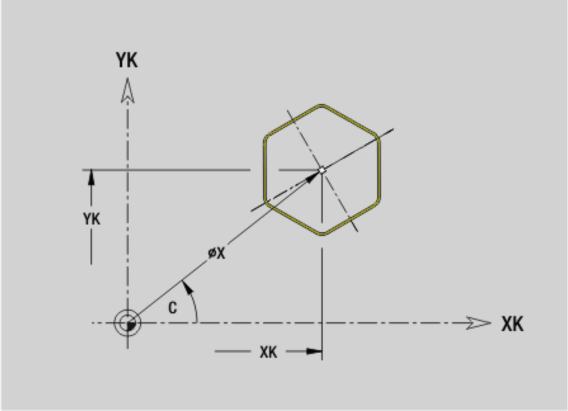
- R>0: Radius of rounding arc
- R<0: Chamfer width

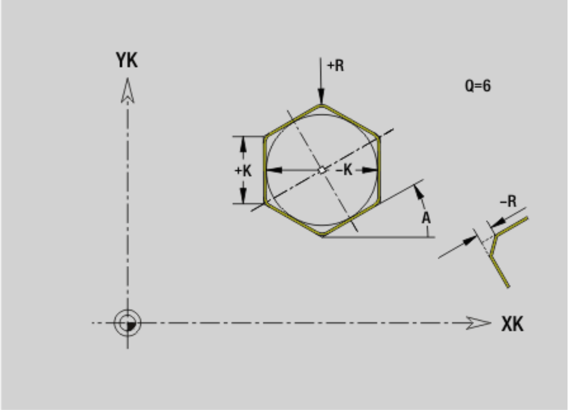
P Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island

Eccentric polygon on front/rear face G307-Geo

G307 defines a polygon in a contour on the front face/rear face.





Parameters

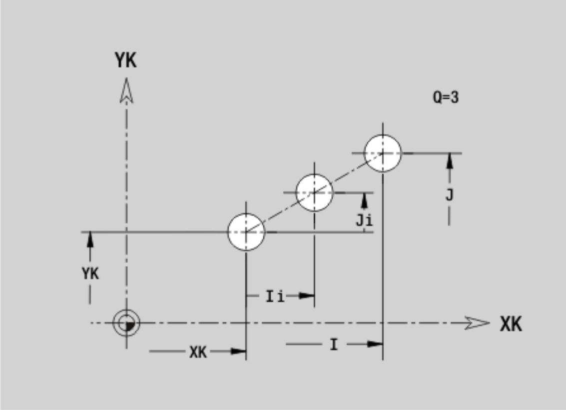
XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
A Angle of a polygon edge to XK axis (default: 0°)
Q Number of edges (Q > 2)
K Edge length
- K>0: Edge length
- K<0: Inside diameter of circle

R Chamfer/rounding arc (default: 0°)
- R>0: Radius of rounding arc
- R<0: Chamfer width

P Depth/height (default: "P" from G308)
- P<0: Pocket
- P>0: Island

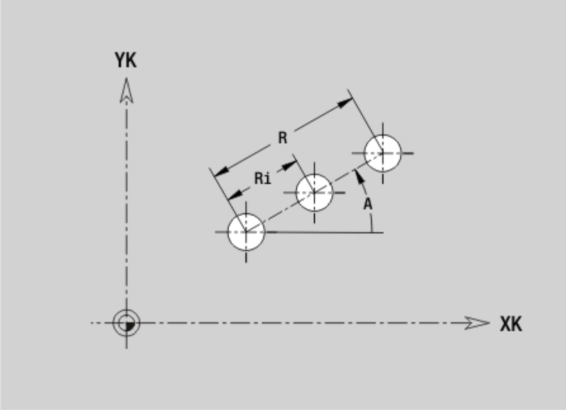
Linear pattern on front/rear face G401-Geo

G401 defines a linear hole pattern or figure pattern on the front or rear face. G401 is effective for the hole/figure defined in the following block (G300 to 305, G307).



Parameters

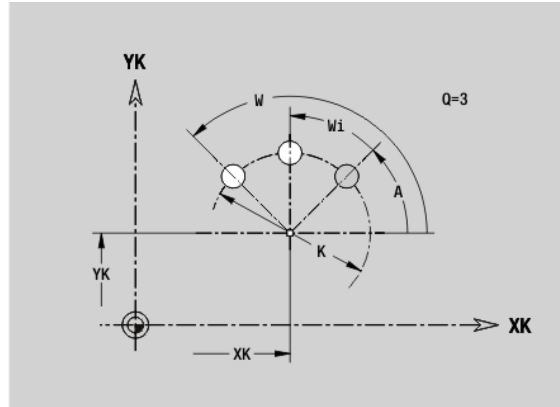
Q	Number of figures (default: 1)
XK	Starting point in Cartesian coordinates
YK	Starting point in Cartesian coordinates
I	End point in Cartesian coordinates
J	End point in Cartesian coordinates
Ii	Distance (XKi) between figures (pattern distance)
Ji	Distance (YKi) between figures (pattern distance)
A	Angle of longitudinal axis to XK axis (default: 0°)
R	Total length of pattern
Ri	Distance between figures (pattern distance)



- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

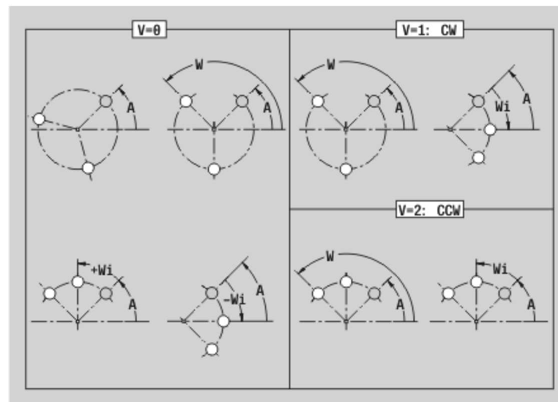
Circular pattern on front/rear face G402-Geo

G402 defines a circular hole pattern or figure pattern on the front or rear face. G402 is effective for the hole/figure defined in the following block (G300 to 305, G307).

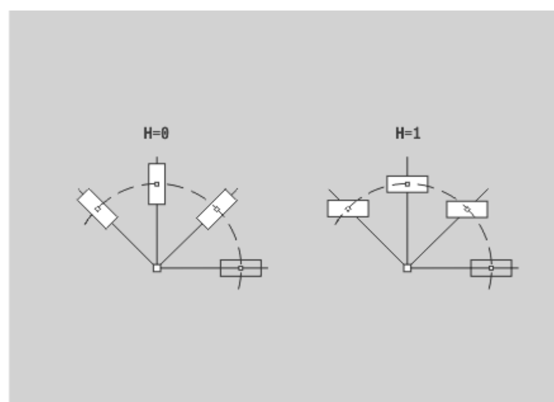


Parameters

- Q — Number of figures
- K — Pattern diameter
- A — Starting angle – position of the first figure; reference: XK axis; (default: 0°)
- W — End angle – position of the last figure; reference: XK axis; (default: 360°)
- Wi — Angle between figures
- V — Direction—orientation (default: 0)
  - V=0, without W: Figures are arranged on a full circle
  - V=0, with W: Figures are arranged on the longer circular arc
  - V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
  - V=1, with W: Clockwise
  - V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
  - V=2, with W: Counterclockwise
  - V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)
- XK — Center in Cartesian coordinates
- YK — Center in Cartesian coordinates
- H — Position of the figures (default: 0)
  - H=0: Normal position; the figures are rotated about the circle center (rotation)
  - H=1: Original position; the position of the figures relative to the coordinate system remains unchanged (translation)



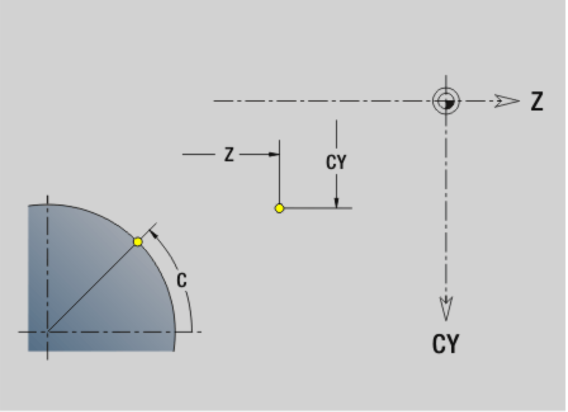
- Program the hole/figure in the following block without a center. Exception: **circular slot**: See "Circular pattern with circular slots" on page 220..
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.



4.8 Lateral surface contours

Starting point of lateral surface contour G110-Geo

G110 defines the starting point of a lateral-surface contour.



Parameters

Z	Start point
C	Starting point (starting angle)
CY	Starting point as linear value; reference: unrolled reference diameter

Program either Z, C or Z, CY.

Line segment in a lateral surface contour G111-Geo

G111 defines a line segment in a lateral-surface contour.

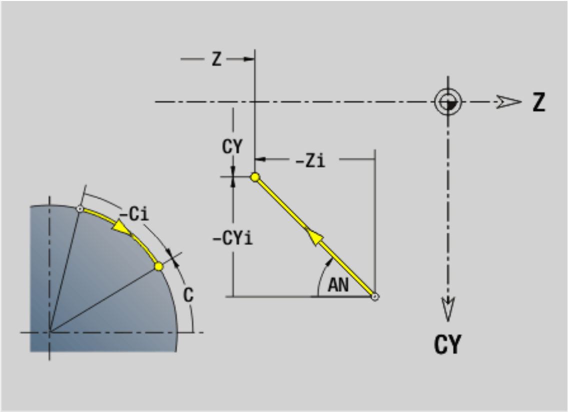
Parameters

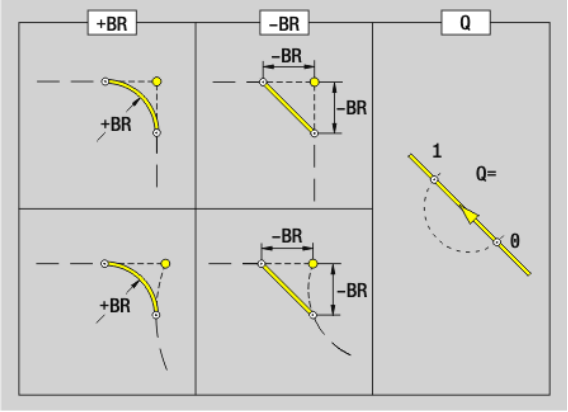
Z	End point
C	End point (end angle)
CY	End point as linear value; reference: unrolled reference diameter
AN	Angle to Z axis
BR	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

Q	Point of intersection. End point if the line segment intersects a line (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection





Programming

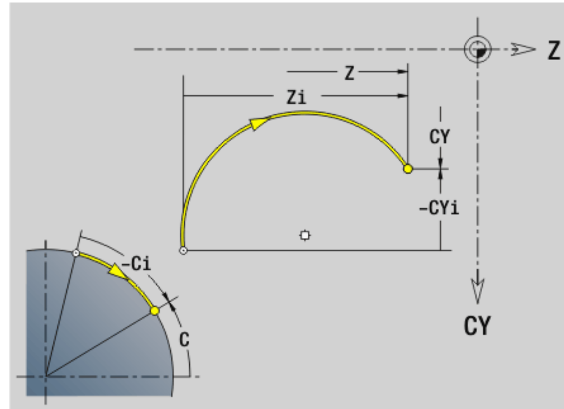
- **Z, CY:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- Program either Z–C or Z–CY

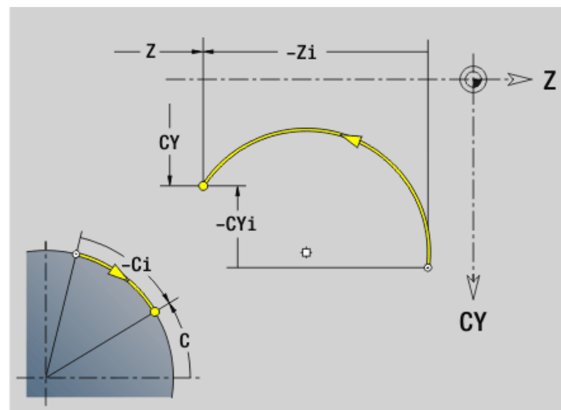
Circular arc in lateral surface contour G112-/ G113-Geo

G112/G113 defines a circular arc in a lateral-surface contour. Direction of rotation: See help graphic

Parameters

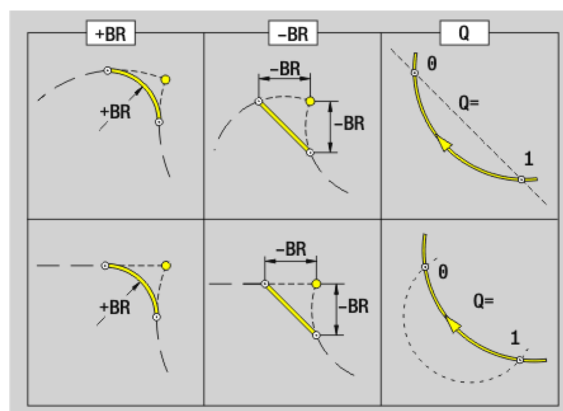
- Z End point
- C End point (end angle)
- CY End point as linear value; reference: unrolled reference diameter
- R Radius
- K Center point in Z direction
- W Angle of the center point
- J Angle of the center point as a linear value
- BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
  - No entry: Tangential transition
  - BR=0: No tangential transition
  - BR>0: Rounding radius
  - BR<0: Width of chamfer
- Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):
  - 0: Near point of intersection
  - 1: Far point of intersection





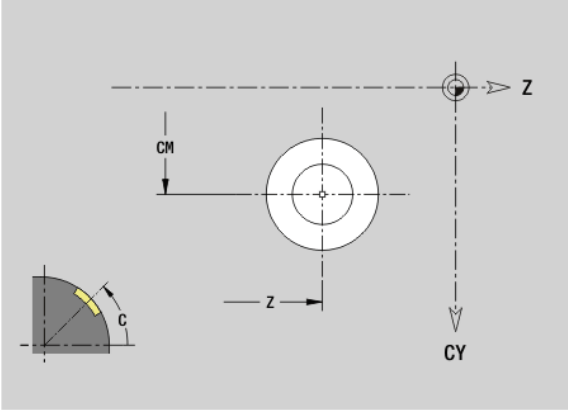
Programming

- **Z, CY:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- **K, J:** Absolute or incremental
- Program either Z–C or Z–CY, and either K–W or K–J
- Program either center or radius
- For radius: Only arcs <= 180° are possible



Hole on lateral surface G310-Geo

G310 defines a hole with countersink and thread in a lateral surface contour.

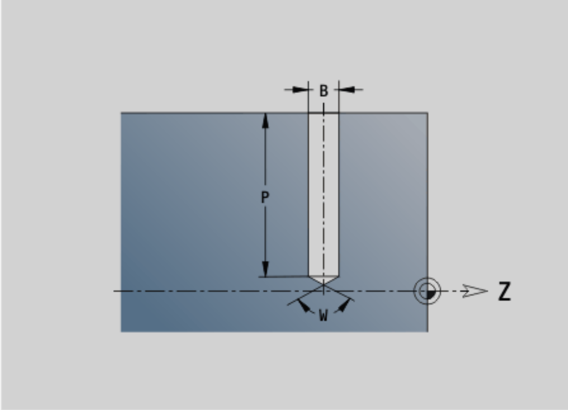


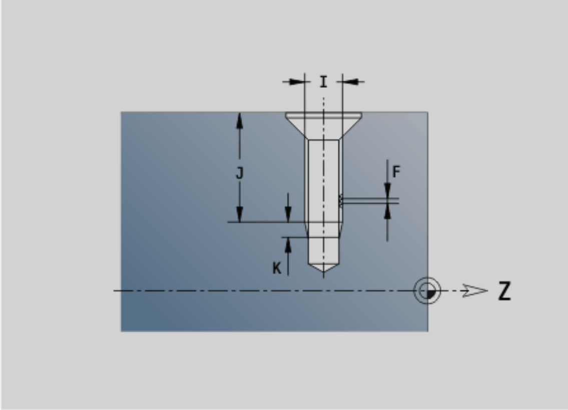
Parameters

Z Center (Z position)
CY Center as linear value; reference: unrolled reference diameter
C Center (angle)
B Hole diameter
P Depth of hole (excluding point)
W Point angle (default: 180°)
R Sinking diameter
U Sinking depth
E Sinking angle
I Thread diameter
J Thread depth
K Thread runout length
F Thread pitch
V Left-hand or right-hand thread (default: 0)

- V=0: Right-hand thread
- V=1: Left-hand thread

A Angle to Z axis; range: 0° < A < 180°; (default: 90° = vertical hole)
O Centering diameter

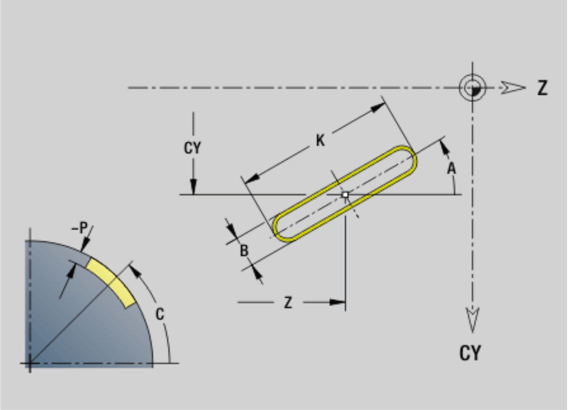




Machine the G310 holes with G71...G74.

Linear slot on lateral surface G311-Geo

G311 defines a linear slot in a lateral-surface contour.



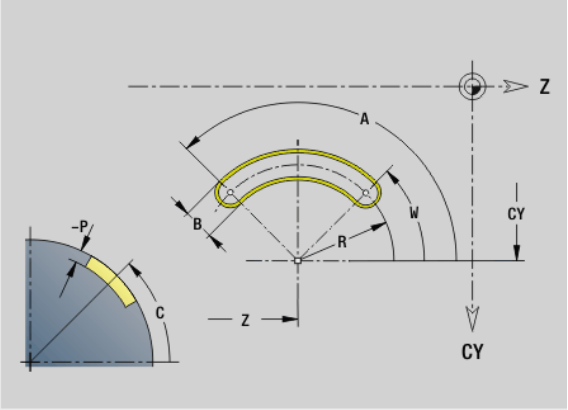
Parameters

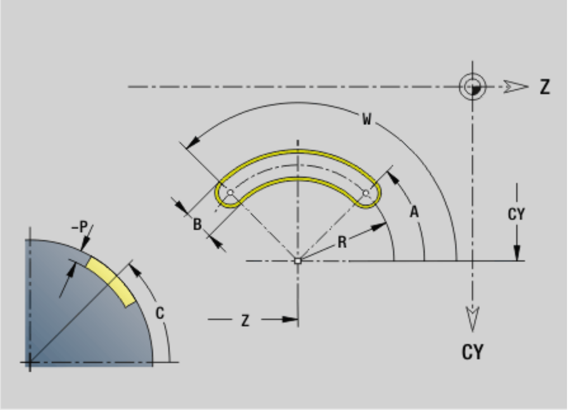
Z	Center (Z position)
CY	Center as linear value; reference: unrolled reference diameter
C	Center (angle)
A	Angle to Z axis (default: 0°)
K	Slot length
B	Slot width
P	Pocket depth (default: "P" from G308)

Circular slot on lateral surface G312/G313-Geo

G312/G313 defines a circular slot in a lateral-surface contour.

- G312: Circular slot clockwise
- G313: Circular slot counterclockwise





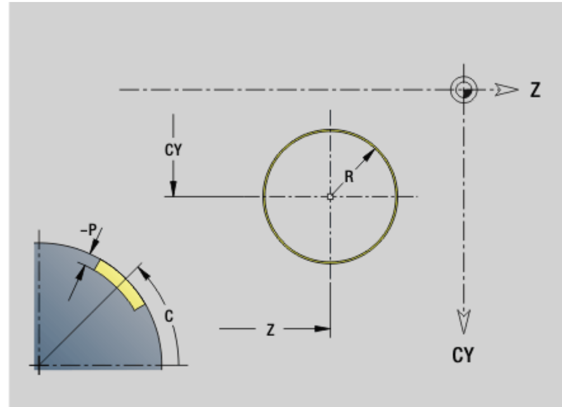
Parameters

Z	Center
CY	Center as linear value; reference: unrolled reference diameter
C	Center (angle)
R	Radius; reference: center point path of the slot
A	Starting angle; reference: Z axis (default: 0°)
W	End angle; reference: Z axis
B	Slot width
P	Pocket depth (default: "P" from G308)

Full circle on lateral surface G314-Geo

G314 defines a full circle in a lateral-surface contour.

Parameters
Z Center
CY Center as linear value; reference: unrolled reference diameter
C Center (angle)
R Radius
P Pocket depth (default: "P" from G308)



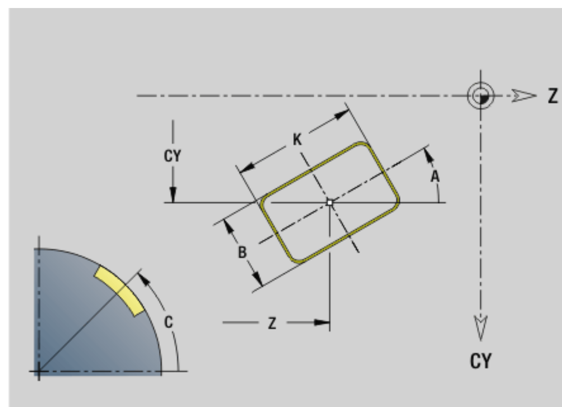
Rectangle on lateral surface G315-Geo

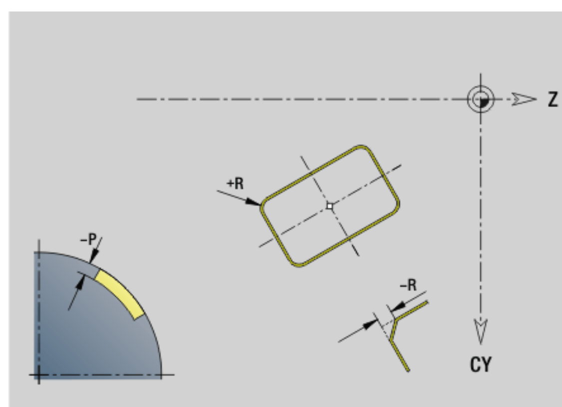
G315 defines a rectangle in a lateral-surface contour.

Parameters
Z Center
CY Center as linear value; reference: unrolled reference diameter
C Center (angle)
A Angle to Z axis (default: 0°)
K Length
B Width
R Chamfer/rounding arc (default: 0°)

- R>0: Radius of rounding arc
- R<0: Chamfer width

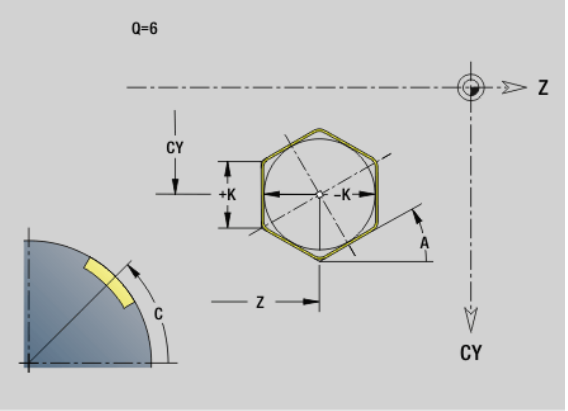
P Pocket depth (default: "P" from G308)





Eccentric polygon on lateral surface G317-Geo

G317 defines a polygon in a lateral-surface contour.



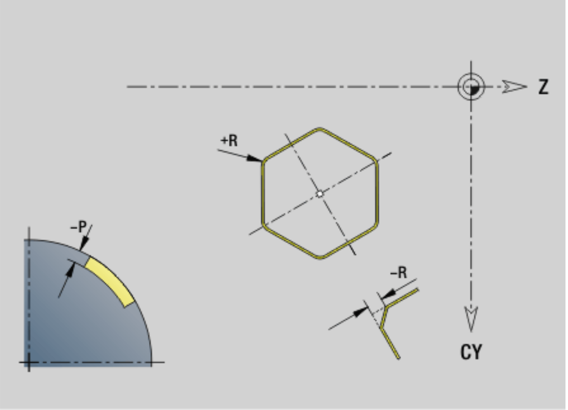
Parameters
Z Center
CY Center as linear value; reference: unrolled reference diameter
C Center (angle)
Q Number of edges (Q > 2)
A Angle to Z axis (default: 0°)
K Edge length

- K>0: Edge length
- K<0: Inside diameter of circle

R Chamfer/rounding arc (default: 0°)

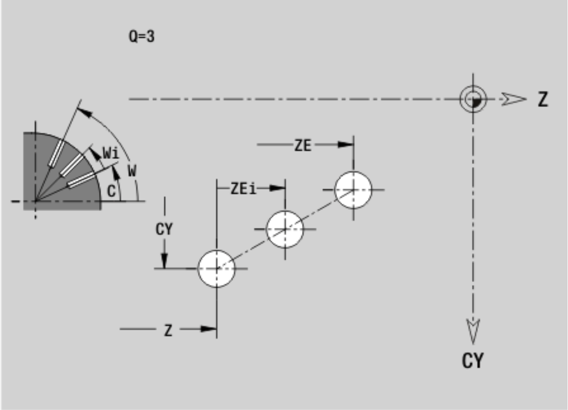
- R>0: Radius of rounding arc
- R<0: Chamfer width

P Pocket depth (default: "P" from G308)



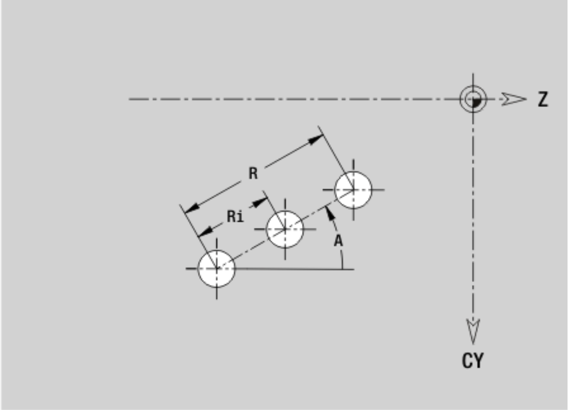
Linear pattern on lateral surface G411-Geo

G411 defines a linear hole or figure pattern on the lateral surface. G411 is effective for the hole/figure defined in the following block (G310 to 315, G317).



Parameters

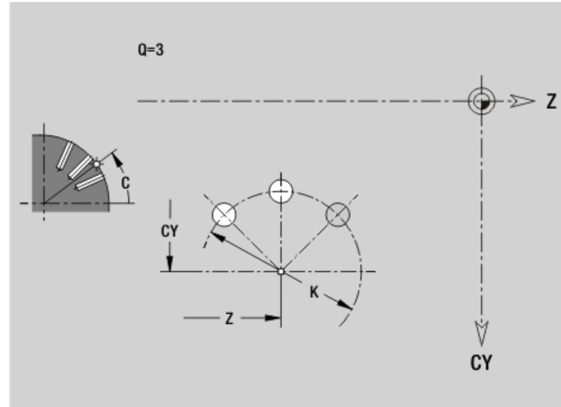
Q	Number of figures (default: 1)
Z	Start point
C	Starting point (starting angle)
CY	Starting point as linear value; reference: unrolled reference diameter
ZE	End point
ZEi	Distance between figures in Z direction
W	End point (end angle)
Wi	Angular distance between figures
A	Angle to Z axis; (default: 0°)
R	Total length of pattern
Ri	Distance between figures (pattern distance)



- If you program Q, Z and C, the holes/figures will be ordered in a regular manner along the circumference.
- Program the hole/figure in the following block without a center.
- The milling cycle calls the hole/figure in the following block—not the pattern definition.

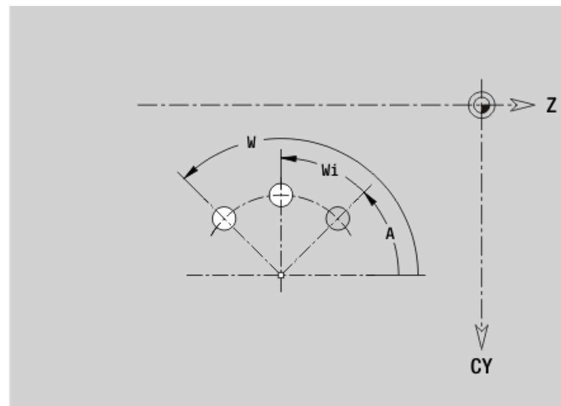
Circular pattern on lateral surface G412-Geo

G412 defines a circular hole or figure pattern on the lateral surface. G412 is effective for the hole/figure defined in the following block (G310 to 315, G317).

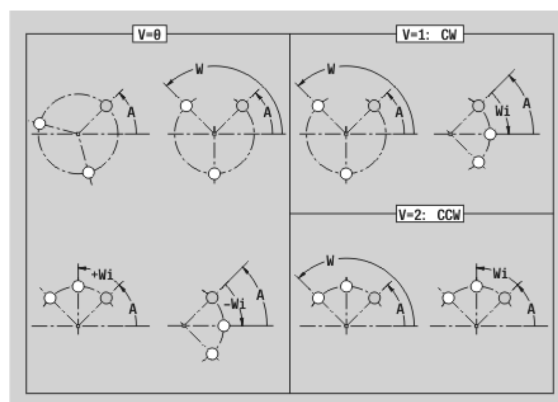


Parameters

- Q Number of figures
- K Pattern diameter
- A Starting angle – position of the first figure; reference: Z axis; (default: 0°)
- W End angle – position of the last figure; reference: Z axis (default: 360°)
- Wi Angle between figures
- V Direction—orientation (default: 0)
  - V=0, without W: Figures are arranged on a full circle
  - V=0, with W: Figures are arranged on the longer circular arc
  - V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
  - V=1, with W: Clockwise
  - V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
  - V=2, with W: Counterclockwise
  - V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)
- Z Center of pattern
- C Center of pattern (angle)
- H Position of the figures (default: 0)
  - H=0: Normal position; the figures are rotated about the circle center (rotation)
  - H=1: Original position; the position of the figures relative to the coordinate system remains unchanged (translation)



- Program the hole/figure in the following block without a center. Exception: **circular slot**: See "Circular pattern with circular slots" on page 220..
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

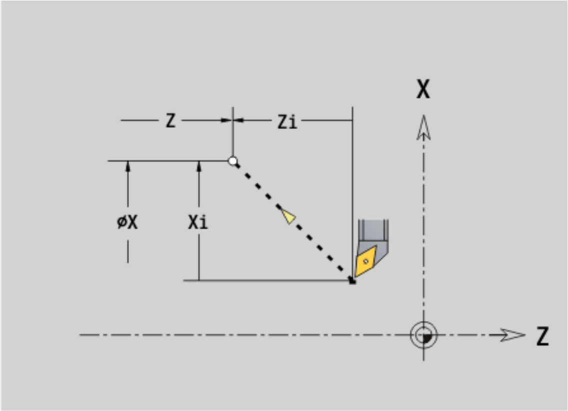


4.9 Tool positioning

Rapid traverse G0

G0 moves at rapid traverse along the shortest path to the target point.

Parameters
X Target point (diameter)
Z Target point



Programming X, Z: Absolute, incremental or modal

If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Rapid traverse to machine coordinates G701

G701 moves at rapid traverse along the shortest path to the target point.

Parameters
X End point (diameter)
Z End point

X, Z refer to the machine zero point and the slide zero point.

If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Setting the tool change position G14

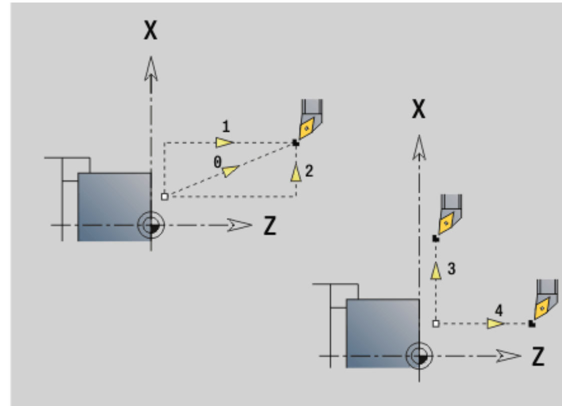
G14 moves the slide at rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change position.

Parameters

Q Sequence. Determines the course of traverse movements (default: 0)

- 0: Diagonal path of traverse
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: Only X direction, Z remains unchanged
- 4: Only Z direction, X remains unchanged

D Number of the tool change position to be approached (0-2) (default =0, tool change position from parameters)



Example: G14

```
. . .
N1 G14 Q0 [Move to the tool change position]
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
. . .
```

Definition of tool-change point G140

G140 defines the position of the tool change point defined in D. This position can be approached with G14.

Parameters

D Number of the tool change point (1-2)

X Diameter—Position of the tool change point

Z Length—Position of the tool change point

> If X or Z parameters are missing, the values from the tool change point parameter are entered.

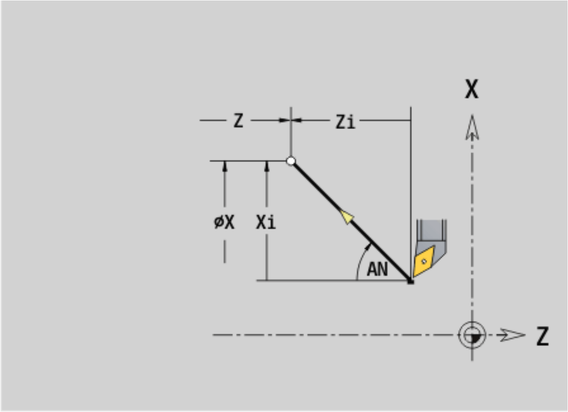
Example: G140

```
. . .
N1  G14 Q0 [Tool change position from parameter]
N2 T3 G95 F0.25 G96 S200 M3
N3 G0  X40 Z10
N5  G140 D1 X100 Z100 [Set tool change pos. 1]
N6  G14 Q0 D1   [Move to tool change pos. 1]
N7  G140 D2 X150 [Set tool change pos. 2, use Z from parameters]
N8  G14 Q0 D2  [Move to tool change pos. 2]
. . .
```

4.10 Linear and circular movements

Linear movement G1

G1 moves the tool on a linear path at the feed rate to the "end point."



Parameters

X End point (diameter)

Z End point

AN Angle (angular direction: see help graphic)

Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

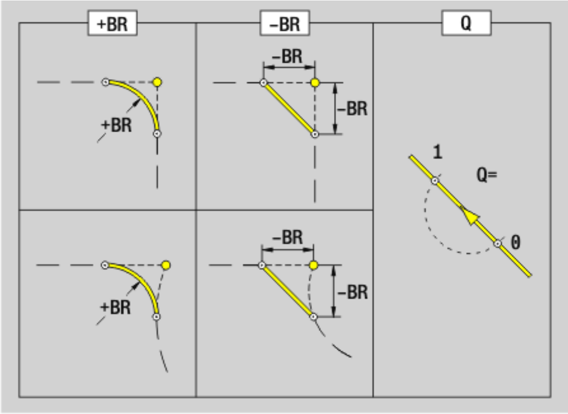
- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

BE Special feed factor for chamfer/rounding arc (default: 1)

Special feed rate = active feed rate * BE (0 < BE <= 1)



Programming X, Z: Absolute, incremental, modal or "?"

If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Circular path G2/ G3

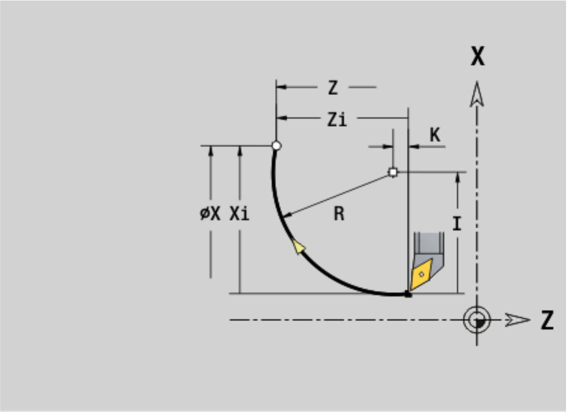
G2/G3 moves the tool in a circular arc at the feed rate to the "end point." The center dimensioning is **incremental.** Direction of rotation (see help graphic):

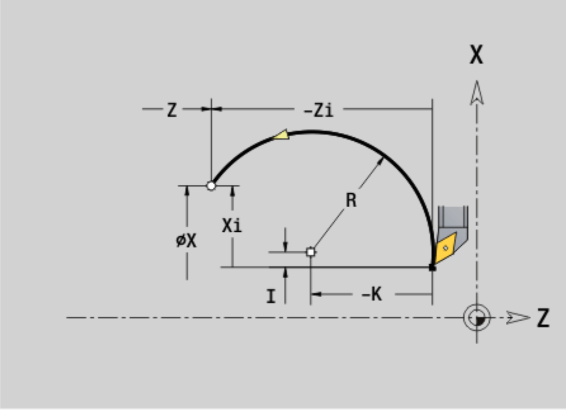
- G2: In clockwise direction
- G3: In counterclockwise direction

Parameters

X	End point (diameter)
Z	End point
R	Radius (0 < R <= 200 000 mm)
I	Incremental center point (distance from starting point to center point; radius)
K	Incremental center point (distance from starting point to center)
Q	Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0): ■ 0: Near point of intersection ■ 1: Far point of intersection
BR	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point. ■ No entry: Tangential transition ■ BR=0: No tangential transition ■ BR>0: Rounding radius ■ BR<0: Width of chamfer
BE	Special feed factor for chamfer/rounding arc (default: 1) Special feed rate = active feed rate * BE (0 < BE <= 1)

Programming X, Z: Absolute, incremental, modal or "?"





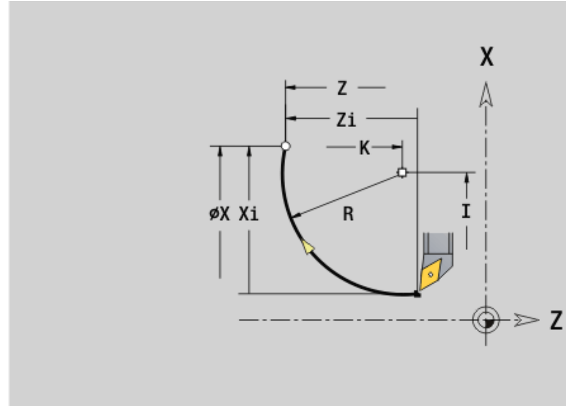
Example: G2, G3

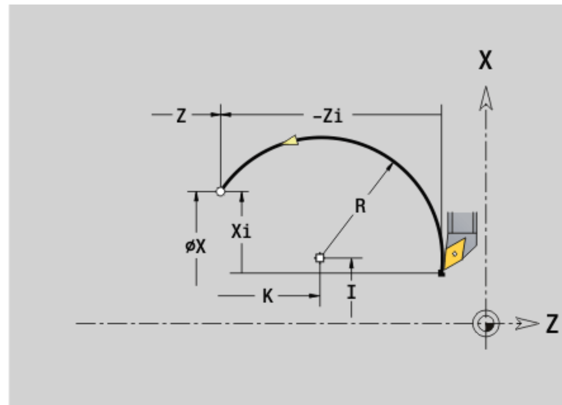
```
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X15 B-0.5 E0.05
N6 G1 Z-25 B0
N7 G2 X45 Z-32 R36 B2
N8 G1 A0
N9 G2 X80 Z-80 R20 B5
N10 G1 Z-95 B0
N11 G3 X80 Z-135 R40 B0
N12 G1 Z-140
N13 G1 X82 G40
. . .
```

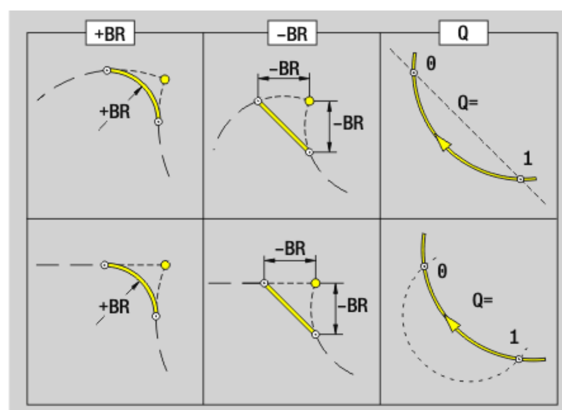
Circular path G12/G13

G12/G13 moves the tool in a circular arc at the feed rate to the "end point." The center dimensioning is **absolute.** Direction of rotation (see help graphic):

- G12: In clockwise direction
- G13: In counterclockwise direction







Parameters

X End point (diameter)

Z End point

R Radius (0 < R <= 200 000 mm)

I Absolute center point (radius)

K Absolute center point

Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

BE Special feed factor for chamfer/rounding arc (default: 1)

Special feed rate = active feed rate * BE (0 < BE <= 1)

Programming X, Z: Absolute, incremental, modal or "?"

4.11 Feed rate, shaft speed

Speed limitation G26

G26: Main spindle; Gx26: Spindle x (x: 1...3)

The speed limitation remains in effect until the end of the program or until a new value is programmed for G26/Gx26.

Parameter
S (Maximum) speed

If S > "absolute maximum speed" (machine parameter), the parameter value will apply.

Example: G26

```
. . .
N1 G14 Q0
N1 G26 S2000 [maximum speed]
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
. . .
```

Interrupted feed G64

G64 interrupts the programmed feed for a short period of time. G64 is a modal function.

Parameters
E Pause duration (0.01 s < E < 99.99 s)
F Feed duration (0.01 s < E < 99.99 s)

- For switch-on, program G64 with E and F.
- For switch-off, program G64 without parameters.

Example: G64

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G64 E0.1 F1 [interrupted feed on]
N3 G0 X0 Z2
N4 G42
N5 G1 Z0
N6 G1 X20 B-0.5
N7 G1 Z-12
N8 G1 Z-24 A20
N9 G1 X48 B6
N10 G1 Z-52 B8
N11 G1 X80 B4 E0.08
N12 G1 Z-60
N13 G1 X82 G40
N14 G64 [interrupted feed off]
. . .
```

Feed per tooth Gx93

Gx93 (x: spindle 1...3) defines the **drive-dependent** feed rate with respect to the number of teeth of the cutter.

Parameter
F Feed per tooth in mm/tooth or inch/tooth

The actual value display shows the feed rate in mm/rev.

Example: G193

```
. . .
N1 M5
N2 T1 G197 S1010 G193 F0.08 M104
N3 M14
N4 G152 C30
N5 G110 C0
N6 G0 X122 Z-50
N7 G...
N8 G...
N9 M15
. . .
```

Constant feed rate G94 (feed per minute)

G94 defines the feed rate **independent of drive.**

Parameter
F Feed per minute in mm/min or in./min

Example: G94

```
. . .
N1 G14 Q0
N2 T3 G94 F2000 G97 S1000 M3
N3 G0 X100 Z2
N4 G1 Z-50
. . .
```

Feed per revolution Gx95

G95: Main spindle; Gx95: Spindle x (x: 1...3)

Gx95 defines a **drive-dependent** feed rate.

Parameter
F Feed rate in mm/revolution or inch/revolution

Example: G95, Gx95

```
. . .
N1 G14 Q0
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
. . .
```

Constant surface speed Gx96

G96: Main spindle; Gx96: Spindle x (x: 1...3)

The spindle speed is dependent on the X position of the tool tip or on the diameter of the drilling or milling tool.

Parameter
S Cutting speed in m/min or ft/min

If you call a drilling tool while a constant cutting speed is active, the Control automatically calculates the spindle speed from the programmed cutting speed and activates it with Gx97. To prevent inadvertent rotation of the spindle, program the **spindle speed first** and **then T**.

Example: G96, G196

```
. . .
N1 T3 G195 F0.25 G196 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20 B-0.5
N6 G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40
. . .
```

Speed Gx97

G97: Main spindle; Gx97: Spindle x (x: 1...3)

Constant spindle speed.

Parameter
S Speed in revolutions per minute

G26/Gx26 limits the spindle speed.

Example: G97, G197

```
. . .
N1 G14 Q0
N2 T3 G95 F0.25 G97 S1000 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
. . .
```

4.12 Tool-tip and cutter radius compensation

Tool-tip radius compensation (TRC)

If TRC is not used, the theoretical tool tip is the reference point for the paths of traverse. This might lead to inaccuracies when the tool moves along non-paraxial paths of traverse. The TRC function corrects programmed paths of traverse.

The TRC (Q=0) **reduces** the feed rate for circular arcs if the shifted radius < the original radius. The TRC corrects the special feed rate when a rounding arc is machined as transition to the next contour element.

Reduced feed rate = feed rate * (shifted radius / original radius)

Milling cutter radius compensation (MCRC)

When the MCRC function is not active, the system defines the center of the cutter as the zero point for the paths of traverse. With the MCRC function, the Control accounts for the outside diameter of the tool when moving along the programmed paths of traverse. The **recessing, roughing and milling cycles** already include TRC/MCRC calls. The TRC/MCRC must be switched off when these cycles are called.

- If the tool radii are > than the contour radii, the TRC/MCRC might cause endless loops. **Recommendation:** Use the finishing cycle G890 or milling cycle G840.
- Never program the MCRC during a perpendicular approach to the machining plane.

G40: Switch off TRC/MCRC

G40 is used to deactivate TRC/MCRC. Please note:

- The TRC/MCRC remains in effect until a block with G40 is reached.
- The block containing G40, or the block after G40 only permits a linear path of traverse (G14 is not permissible).

Function of the TRC/MCRC

. . .	
N.. G0 X10 Z10	
N.. G41	Activate TRC to the left of the contour
N.. G0 Z20	Path of traverse: from X10/Z10 to X10+TRC/Z20+TRC
N.. G1 X20	The path of traverse is "shifted" by the TRC
N.. G40 G0 X30 Z30	Path of traverse from X20+TRC/Z20+TRC to X30/Z30
. . .	

G41/G42: Switch on TRC/MCRC

G41: Switch on TRC/MCRC—compensation of the tool-tip/cutter radius to the **left** of the contour in traverse direction.

G42: Switch on TRC/MCRC—compensation of the tool-tip/cutter radius to the **right** of the contour in traverse direction.

Parameters

Q Plane (default: 0)

- 0: TRC on the turning plane (XZ plane)
- 1: MCRC on the front face (XC plane)
- 2: MCRC on the lateral surface (ZC plane)
- 3: MCRC on the front face (XY plane)
- 4: MCRC on the lateral surface (YZ plane)

H Output (only with TRC) – (default: 0)

- 0: Intersecting areas which are programmed in directly successive contour elements are not machined.
- 1: The complete contour is machined—even if certain areas are intersecting.

O Feed rate reduction (default: 0)

- 0: Feed rate reduction is active
- 1: No feed rate reduction

Please note:

- Program G41/G42 in a separate NC block.
- Program a straight line segment (G0/G1) after the block containing G41/G42.
- The TRC/MCRC is taken into account from the next path of traverse.

Example: G40, G41, G42

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42 [TRC on, to the right of the contour]
N4 G1 Z0
N5 G1 X20 B-0.5
N6 G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G4 [TRC off]
. . .
```

4.13 Zero point shifts

You can program several zero shifts in one NC program. The relationships of the coordinates (for blank/finished part, auxiliary contours) are retained by the zero offset description.

G920 temporarily deactivates zero point shifts—G980 reactivates them.

Overview of zero point shifts	
G51: ■ Relative shift ■ Programmed shift ■ Reference: Previously defined workpiece zero point	Page 252
G56: ■ Additive shift ■ Programmed shift ■ Reference: Workpiece zero point defined at present	Page 253
G59: ■ Absolute shift ■ Programmed shift ■ Reference: Machine zero point	Page 254

Zero point shift G51

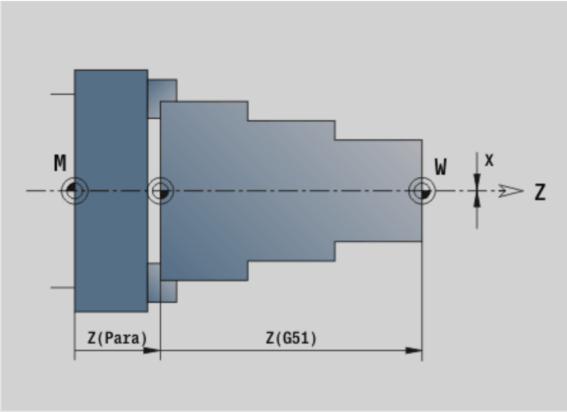
G51 shifts the workpiece zero point by Z (and X). The shift is referenced to the workpiece zero point defined in setup mode.

Parameters

X Shift (radius)
Z Displacement (shift)

Even if you shift the zero point several times with G51, it is always referenced to the workpiece zero point defined in setup mode.

The zero point shift is valid until program end, or until it is canceled by other zero point shifts.



Example: G51

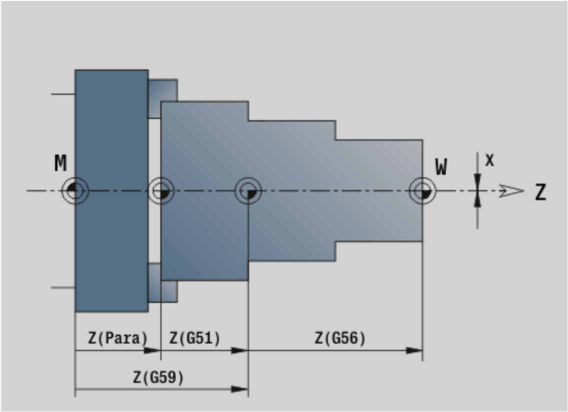
```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X62 Z5
N3 G810 NS7 NE12 P5 I0.5 K0.2
N4 G51 Z-28 [zero point shift]
N5 G0 X62 Z-15
N6 G810 NS7 NE12 P5 I0.5 K0.2
N7 G51 Z-56 [zero point shift]
. . .
```

Additive zero point shift G56

G56 shifts the workpiece zero point by Z (and X). The shift is referenced to the currently active workpiece zero point.

Parameters
X Shift (radius value) – (default: 0)
Z Shift

If you shift the workpiece zero point more than once with G56, the shift is always added to the currently active zero point.



Example: G56

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X62 Z5
N3 G810 NS7 NE12 P5 I0.5 K0.2
N4 G56 Z-28 [zero point shift]
N5 G0 X62 Z5
N6 G810 NS7 NE12 P5 I0.5 K0.2
N7 G56 Z-28 [zero point shift]
. . .
```

Absolute zero point shift G59

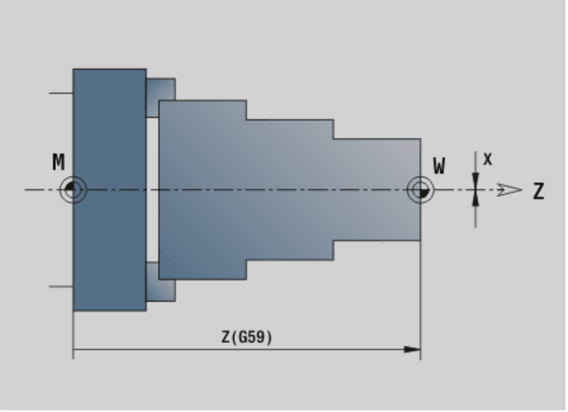
G59 sets the workpiece zero point to X, Z. The new zero point remains in effect to the end of the program.

Parameters

X Shift (radius)

Z Shift

G59 cancels all previous zero point shifts (with G51, G56 or G59).



Example: G59

```
. . .
N1 G59 Z256 [zero point shift]
N2 G14 Q0
N3 T3 G95 F0.25 G96 S200 M3
N4 G0 X62 Z2
. . .
```

4.14 Oversizes

Switch off oversize G50

G50 switches off oversizes defined with G52-Geo for the following cycle. Program G50 before the cycle.

To ensure compatibility, the G52 code is also supported for switching off the oversizes. HEIDENHAIN recommends using G50 for new NC programs.

Axis-parallel oversize G57

G57 defines different oversizes for X and Z. Program G57 before the cycle call.

Parameters

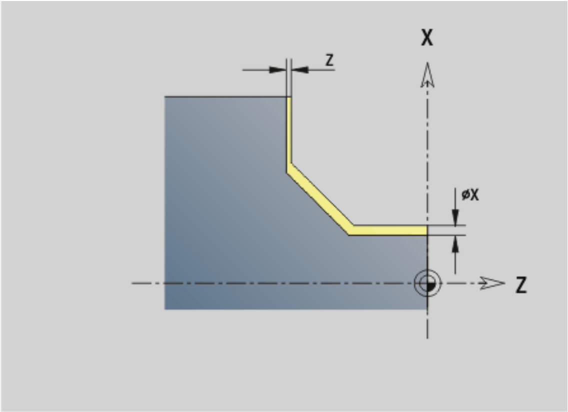
X Oversize X (diameter value) – only positive values

Z Oversize Z – only positive values

G57 is effective in the following cycles. After cycle run, the oversizes are

- Deleted: G810, G820, G830, G835, G860, G869, G890
- **Not** deleted: G81, G82, G83

If the oversizes are programmed with G57 **and** in the cycle itself, the cycle oversizes apply.



Example: G57

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G57 X0.2 Z0.5 [paraxial oversize]
N4 G810 NS7 NE12 P5
. . .
```

Contour-parallel oversize (equidistant) G58

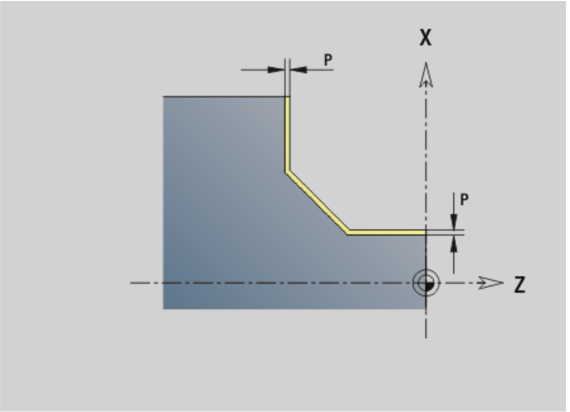
G58 defines an equidistant oversize. Program G58 before the cycle call. A negative oversize during finishing is permitted with G890.

Parameters
P Oversize

G58 is effective in the following cycles. After cycle run, the oversizes are

- deleted: G810, G820, G830, G835, G860, G869, G890
- **not** deleted: G83

If an oversize is programmed with G58 **and** in the cycle, the oversize from the cycle is used.



Example: G58

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G58 P2 [contour-parallel oversize]
N4 G810 NS7 NE12 P5
. . .
```

4.15 Safety clearances

Safety clearance G47

G47 defines the safety clearance for

- the turning cycles: G810, G820, G830, G835, G860, G869, G890.
- the drilling cycles G71, G72, G74.
- the milling cycles G840...G846.

Parameter
P Safety clearance

G47 without parameters activates the parameter values defined in the "Safety clearance G47" user parameter.

G47 replaces the safety clearance set in the machining parameters or that set in G147.

Safety clearance G147

G147 defines the safety clearance for

- the milling cycles G840...G846.
- the drilling cycles G71, G72, G74.

Parameters
I Safety clearance to the milling plane (only for milling operations)
K Safety clearance in approach direction (feed)

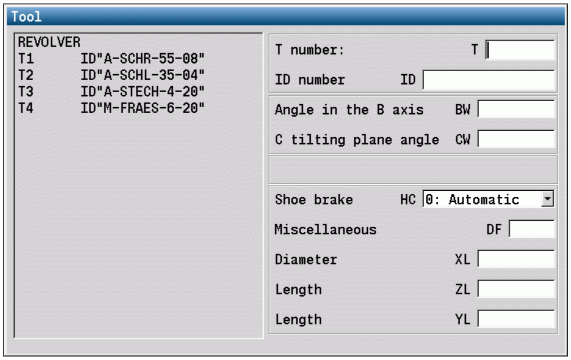
G147 without parameters activates the parameter values defined in the "Safety clearance G147.." user parameter.

G147 replaces the safety clearance set in the machining parameters or that set in G47.

4.16 Tools, compensations

Tool call T

The Control displays the tool assignment defined in the TURRET section. You can enter the T number directly or select it from the tool list (switch with the **Tool list** soft key).



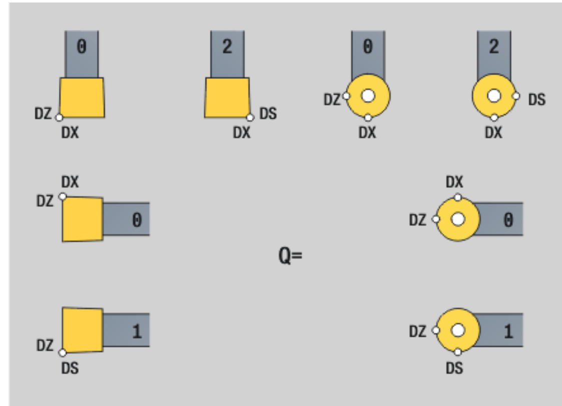
Correction of cut (switching the tool edge compensation) G148

G148 defines the values compensating for wear. DX, DZ become effective after program start and after a T command.

Parameter
Q Selection (default: 0)

- O=0: DX, DZ active—DS inactive
- O=1: DS, DZ active—DX inactive
- O=2: DX, DS active—DZ inactive

> The cycles G860, G869, G879, G870, G890 automatically take the "correct" wear compensation into account.



Example: G148

```
. . .
N1 T3 G95 F0.25 G96 S160 M3
N2 G0 X62 Z2
N3 G0 Z-29.8
N4 G1 X50.4
N5 G0 X62
N6 G150
N7 G1 Z-20.2
N8 G1 X50.4
N9 G0 X62
N10 G151 [recessing finishing]
N11 G148 O0 [change compensation]
N12 G0 X62 Z-30
N13 G1 X50
N14 G0 X62
N15 G150
N16 G148 O2
N17 G1 Z-20
N18 G1 X50
N19 G0 X62
. . .
```

Additive compensation G149

The Control manages 16 tool-independent compensation values. One G149 followed by a D number activates the additive compensation function. G149 D900 deactivates the additive compensation function. The compensation values are managed in the Program Run mode (see "Program Run mode" in the User's Manual).

Parameter

D   Additive compensation (default: D900):

- D900: deactivates the additive compensation
- D901 to D916: activates the additive compensation

Programming:

- The compensation becomes effective after the tool has moved in the compensation direction by the compensation value. Therefore, program G149 one block before the block containing the path of traverse to which the compensation is to apply.
- Additive compensation remains in effect up to:
  - the next G149 D900
  - the next tool change
  - End of program

> The additive compensation is added to the tool compensation.

Example: G149

```
. . .
N1 T3 G96 S200 G95 F0.4 M4
N2 G0 X62 Z2
N3 G89
N4 G42
N5 G0 X27 Z0
N6 G1 X30 Z-1.5
N7 G1 Z-25
N8 G149 D901 [activate compensation]
N9 G1 X40 BR-1
N10 G1 Z-50
N11 G149 D902
N12 G1 X50 BR-1
N13 G1 Z-75
N14 G149 D900 [deactivate compensation]
N15 G1 X60 B-1
N16 G1 Z-80
N17 G1 X62
N18 G80
. . .
```

Compensation of right-hand tool tip G150
Compensation of left-hand tool tip G151

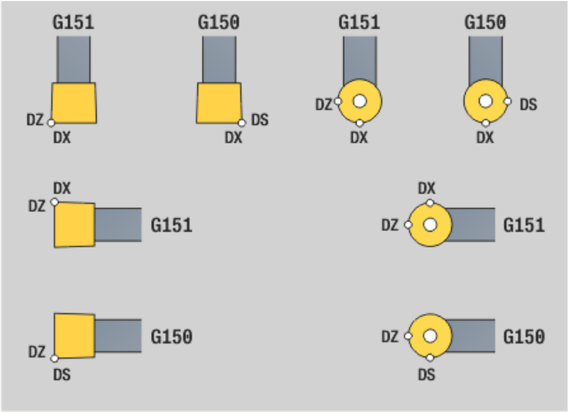
G150/G151 defines the tool reference point for recessing and button tools.

- G150: Reference point is on right tip
- G151: Reference point is on left tip.

G150/G151 is effective from the block in which it is programmed and remains in effect up to

- the next tool change
- program end.

> - The displayed actual values always refer to the tool tip defined in the tool data.
> - If you use TRC, after G150/G151 you must also adjust G41/G42.



Example: G150, G151

```
. . .
N1 T3 G95 F0.25 G96 S160 M3
N2 G0 X62 Z2
N3 G0 Z-29.8
N4 G1 X50.4
N5 G0 X62
N6 G150
N7 G1 Z-20.2
N8 G1 X50.4
N9 G0 X62
N10 G151 [recessing finishing]
N11 G148 O0
N12 G0 X62 Z-30
N13 G1 X50
N14 G0 X62
N15 G150
N16 G148 O2
N17 G1 Z-20
N18 G1 X50
N19 G0 X62
. . .
```

4.17 Contour-based turning cycles

Working with contour-based cycles

Possibilities of transferring the contour to be machined to the cycle:

- Transferring the contour reference in the start block number and the end block number. The contour area is machined in the direction "from NS to NE."
- Transferring the contour reference via the name of the auxiliary contour (ID). The complete auxiliary contour is machined in the direction of contour definition.
- Describing the contour with G80 in the block directly after the cycle (see "Cycle end / Simple contour G80" on page 286).
- Describing the contour with G0, G1, G2 and G3 blocks directly after the cycle. The contour is concluded by G80 without parameters.

Possibilities of defining the workpiece blank for calculating the number of cutting passes:

- Defining a global workpiece blank in the **WORKPIECE BLANK** program section. Regeneration of the workpiece blank is automatically active. The cycle uses the specified workpiece blank.
- If no workpiece blank is defined, the cycle calculates the blank from the contour to be machined and the position of the tool during cycle call. Contour follow-up is **not** active.

Finding the block references:

Contour reference

- Place cursor in NS or NE input field
- Press the soft key

Select the contour element:

- Use the horizontal arrow keys to select the contour element
- Use the vertical arrow keys to switch between contours (also face contours, etc.).

NS

Switch between NS and NE:

- Press the NS soft key
- Press the NE soft key

Take over

- Press the soft key to confirm the block number and return to the dialog.

Example: Contour-based cycles

```
. . .
N1 G810 NS7 NE12 P3[block reference]
N2 ...
N3 G810 ID"007" P3 [name of auxiliary contour]
N4 ...
N5 G810 ID"007" NS9 NE7 P [combination]
N6 ...
N7 G810 P3[predefined contour description]
N8 G80 XS60 ZS-2 XE90 ZE-50 AC10 WC10 BS3 BE-2 RC5 EC0
N9...
N10 G810 P3[direct contour description]
N11 G0 X50 Z0
N12 G1 Z-62 BR4
N13 G1 X85 AN80 BR-2
N14 G1 Zi-5
N15 G80
N16 ...
. . .
```

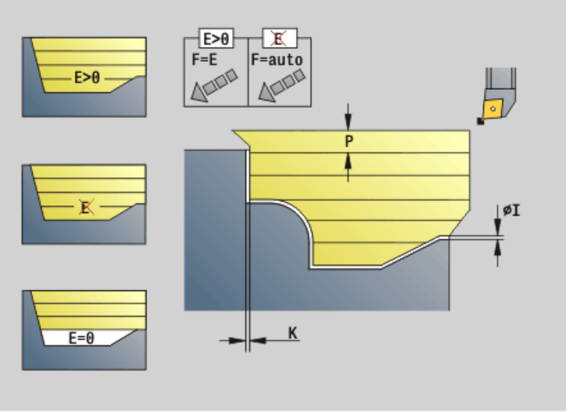
Cutting limits in X, Z

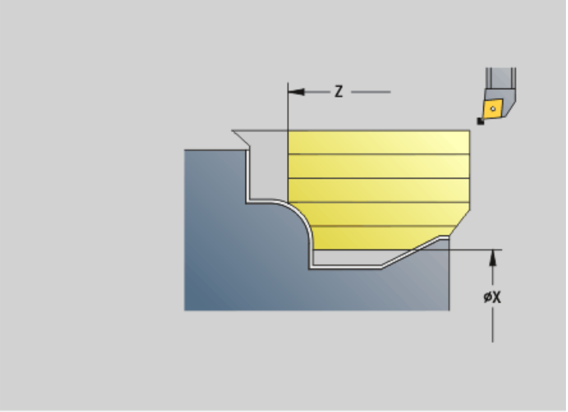
The tool position before the cycle call determines the effect of a cutting limit. The Control machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

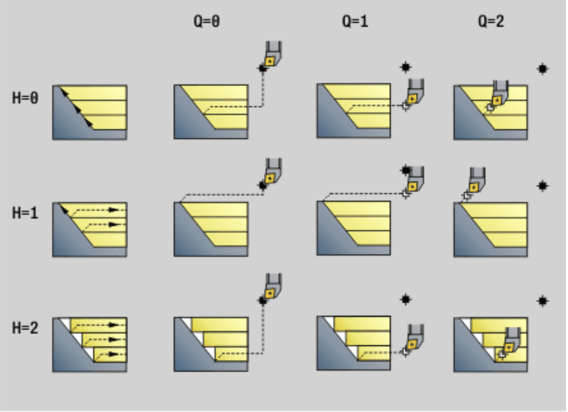
A cutting limit restricts the contour area that can be machined; it does not apply to the paths for approach and departure.

Longitudinal roughing G810

G810 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call (see "Working with contour-based cycles" on page 262). The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.







Parameters

ID Auxiliary contour—ID number of the contour to be machined

NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

P Maximum infeed

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

E Plunging behavior

- E=0: Descending contours are not machined
- E>0: Plunging feed rate
- One input: Feed rate reduction depending on the plunge angle – maximum 50%

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

A Approach angle (reference: Z-axis) – (default: 0°/180°, parallel to Z-axis)

W Departing angle (reference: Z-axis) – (default: 90°/270°; perpendicular to Z-axis)

H Type of departure (default: 0)

- 0: With each cut (machine contour outline after each pass)
- 1: With the last cut (retracts at 45°; contour smoothing after last pass)
- 2: No smoothing (retracts at 45°; no contour smoothing)

Q Type of retraction at cycle end (default: 0)

- 0: Returns to starting point, first X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

Parameters

V Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining
- 4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

D Omit elements (see figure)

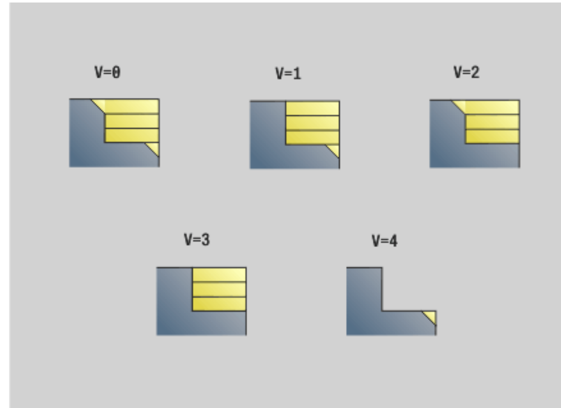
O Hide undercutting:

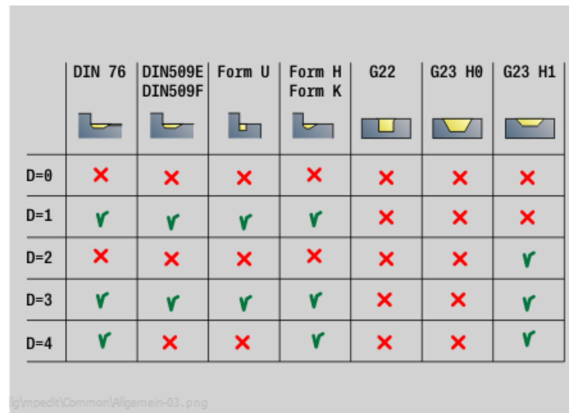
- 0: Undercuts are machined
- 1: Undercuts are not machined

B Slide lead with 4-axis machining (not yet implemented)

XA, ZA Starting point of blank (only effective if no blank was programmed):

- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the workpiece blank.





	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

The Control uses the tool definition to distinguish between external and internal machining.

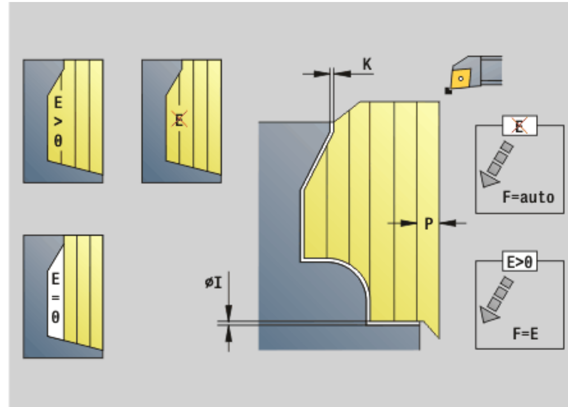
- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

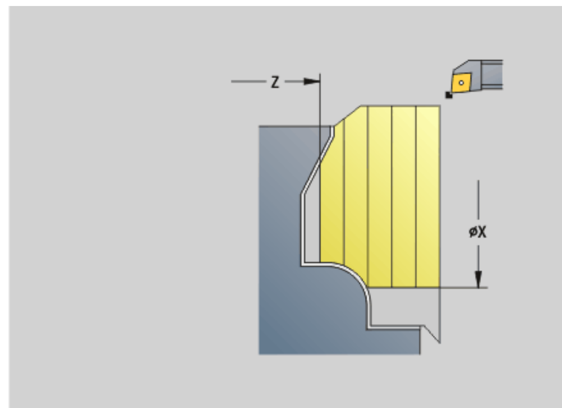
Cycle run

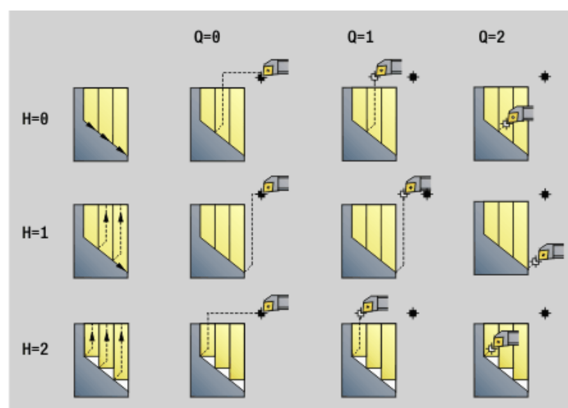
1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account (first in Z direction, then in X).
3. Move at feed rate to target point Z
4. Depending on H:
   - H=0: Machines the contour outline
   - H=1 or 2: Retracts at 45°
5. Returns at rapid traverse and approaches for next pass.
6. Repeats 3 to 5 until target point X has been reached.
7. If required, repeats 2 to 6 until all areas have been machined.
8. If H=1: Smoothes the contour
9. Retracts as programmed in Q.

Face roughing G820

G820 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call (see "Working with contour-based cycles" on page 262). The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.







Parameters

ID Auxiliary contour—ID number of the contour to be machined

NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

P Maximum infeed

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

E Plunging behavior

- E=0: Descending contours are not machined
- E>0: Plunging feed rate
- No input: Feed rate reduction depending on the plunge angle – maximum 50%

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

A Approaching angle (reference: Z-axis) – (default: 90°/270°; perpendicular to Z-axis)

W Departure angle (reference: Z-axis) – (default: 0°/180°, parallel to Z-axis)

H Type of departure (default: 0)

- 0: With each cut (machine contour outline after each pass)
- 1: With the last cut (retracts at 45°; contour smoothing after last pass)
- 2: No smoothing (retracts at 45°; no contour smoothing)

Q Type of retraction at cycle end (default: 0)

- 0: Returns to starting point, first Z, then X direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

Parameters

V Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining
- 4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

D Omit elements (see figure)

O Hide undercutting:

- 0: Undercuts are machined
- 1: Undercuts are not machined

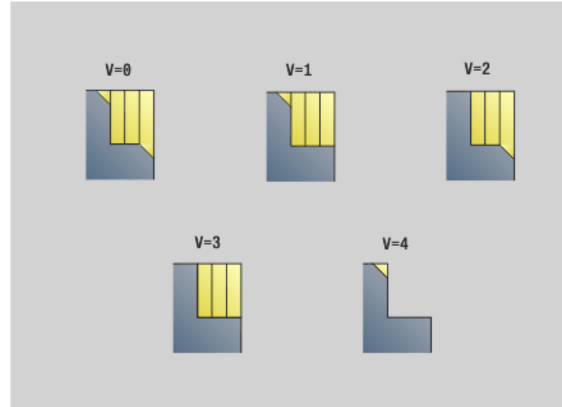
B Slide lead with 4-axis machining (not yet implemented)

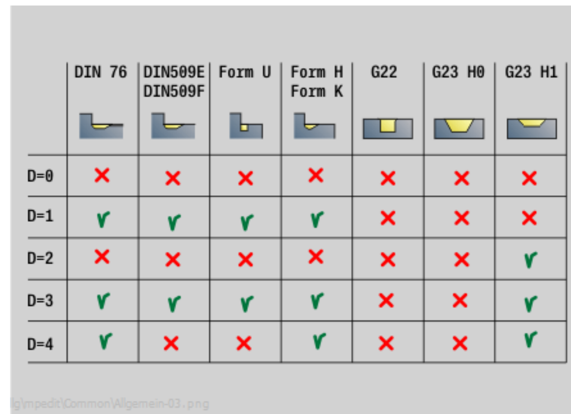
XA, ZA Starting point of blank (only effective if no blank was programmed):

- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the workpiece blank.

The Control uses the tool definition to distinguish between external and internal machining.

- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.





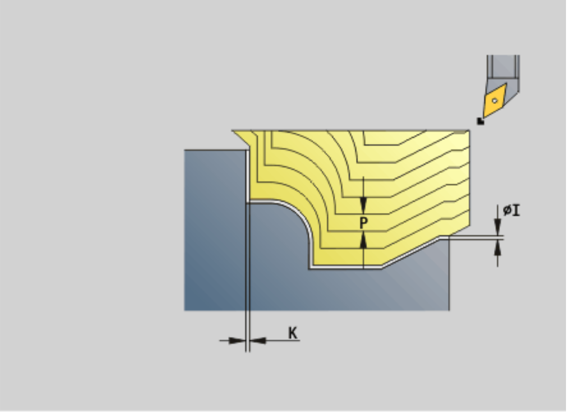
	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

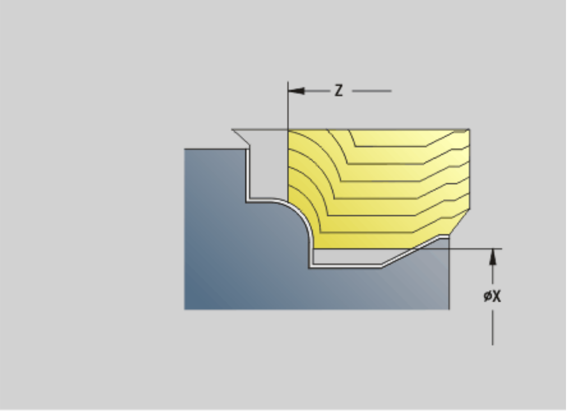
Cycle run

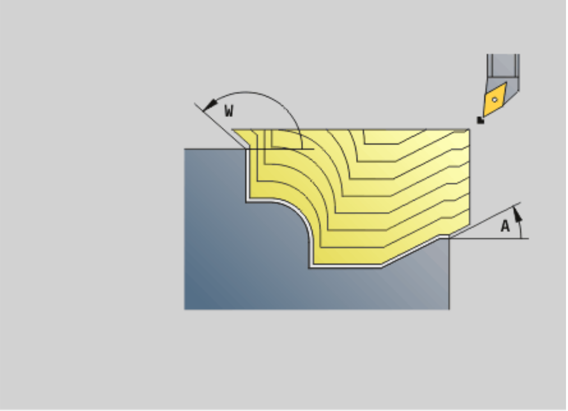
1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account (first in X direction, then in Z).
3. Move at feed rate to target point X
4. Depending on H:
   - H=0: Machines the contour outline
   - H=1 or 2: Retracts at 45°
5. Returns at rapid traverse and approaches for next pass.
6. Repeats 3 to 5 until target point Z has been reached.
7. If required, repeats 2 to 6 until all areas have been machined.
8. If H=1: Smoothes the contour
9. Retracts as programmed in Q.

Contour-parallel roughing G830

G830 machines the contour area defined in "ID", or by "NS, NE", parallel to the contour (see "Working with contour-based cycles" on page 262). The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.







Parameters

ID	Auxiliary contour—ID number of the contour to be machined
NS	Starting block number (beginning of contour section)
NE	End block number (end of contour section) ■ NE not programmed: The contour element NS is machined in the direction of contour definition. ■ NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.
P	Maximum infeed
I	Oversize in X direction (diameter value) – (default: 0)
K	Oversize in Z direction (default: 0)
X	Cutting limit in X direction (diameter value) – (default: no cutting limit)
Z	Cutting limit in Z direction (default: no cutting limit)
A	Approach angle (reference: Z axis)—(default: 0°/180°, parallel to Z axis, or with facing tools: parallel to X axis)
W	Departure angle (reference: Z axis)—(default: 90°/270°, perpendicular to Z axis, or with facing tools: perpendicular to X axis)
Q	Type of retraction at cycle end (default: 0) ■ 0: Returns to starting point, first X, then Z direction ■ 1: Positions in front of the finished contour ■ 2: Retracts to safety clearance and stops

Parameters

V Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining
- 4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

B Contour calculation

- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

D Omit elements (see figure)

J aß (radius value)—active only if **no blank** has been defined.

H Contour-parallel—Type of cutting paths:

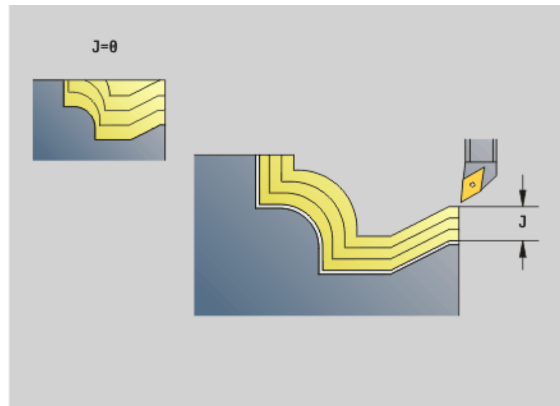
- 0: Constant machining depth
- 1: Equidistant cut lines

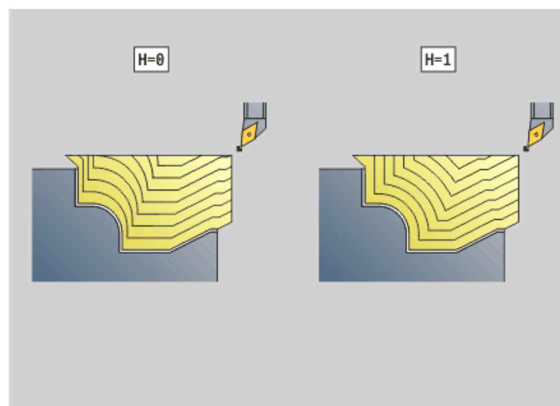
HR Specify primary machining direction

XA, ZA Starting point of blank (only effective if no blank was programmed):

- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the workpiece blank.

	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	×	×	×	×	×	×	×
D=1	✓	✓	✓	✓	×	×	×
D=2	×	×	×	×	×	×	✓
D=3	✓	✓	✓	✓	×	×	✓
D=4	✓	×	×	✓	×	×	✓





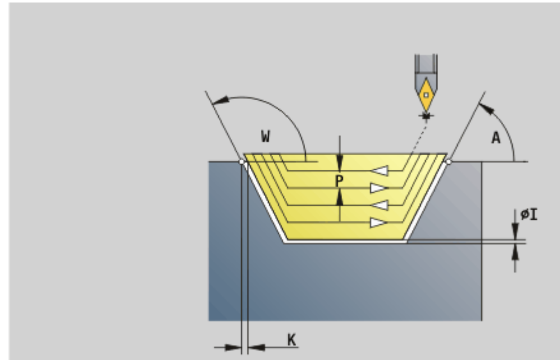
The Control uses the tool definition to distinguish between external and internal machining.

- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account.
3. Executes the first cut (roughing).
4. Returns at rapid traverse and approaches for next pass.
5. Repeats 3 to 4 until the complete area has been machined.
6. If required, repeats 2 to 5 until all areas have been machined.
7. Retracts as programmed in Q.

Contour cycle, bidirectional (contour-parallel with neutral tool) G835



G835 machines the contour area defined in "ID", or by "NS, NE", parallel to the contour and bidirectionally (see "Working with contour-based cycles" on page 262). The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.

Parameters

ID Auxiliary contour—ID number of the contour to be machined
NS Starting block number (beginning of contour section)
NE End block number (end of contour section)

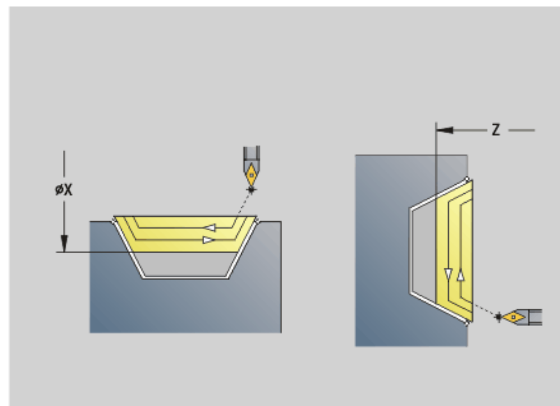
- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

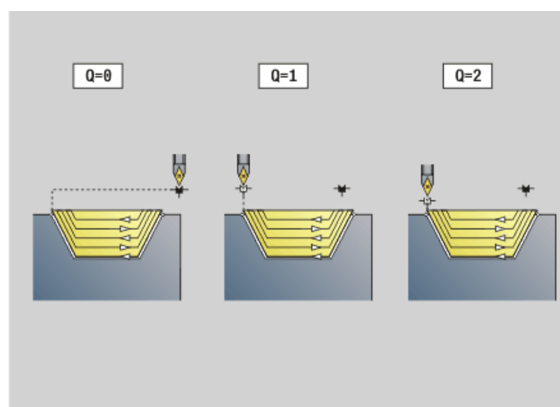
P Maximum infeed
I Oversize in X direction (diameter value) – (default: 0)
K Oversize in Z direction (default: 0)
X Cutting limit in X direction (diameter value) – (default: no cutting limit)
Z Cutting limit in Z direction (default: no cutting limit)
A Approach angle (reference: Z axis)—(default: 0°/180°, parallel to Z axis, or with facing tools: parallel to X axis)
W Departure angle (reference: Z axis)—(default: 90°/270°, perpendicular to Z axis, or with facing tools: perpendicular to X axis)
Q Type of retraction at cycle end (default: 0)

- 0: Returns to starting point (first X, then Z direction)
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

V Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining
- 4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)





Parameters

B Contour calculation
- 0: Automatic
- 1: Tool to the left (G41)
- 2: Tool to the right (G42)

D Omit elements (see figure)

J aß (radius value)—active only if **no blank** has been defined.

H Contour-parallel—Type of cutting paths:
- 0: Constant machining depth
- 1: Equidistant cut lines

XA, ZA Starting point of blank (only effective if no blank was programmed):
- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the workpiece blank.

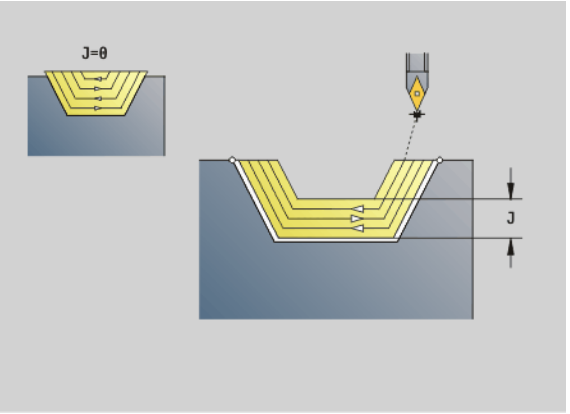
The Control uses the tool definition to distinguish between external and internal machining.

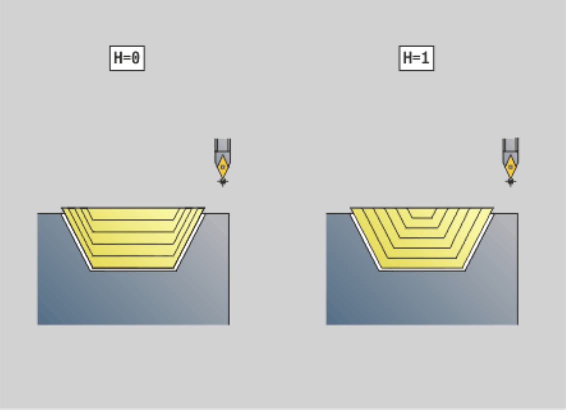
- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run

1 Calculates the areas to be machined and the cutting segmentation.
2 Approaches workpiece for first pass from starting point, taking the safety clearance into account.
3 Executes the first cut (roughing).
4 Approaches for the next pass and execute the next cut (roughing) in the opposite direction.
5 Repeats 3 to 4 until the complete area has been machined.
6 If required, repeats 2 to 5 until all areas have been machined.
7 Retracts as programmed in Q.

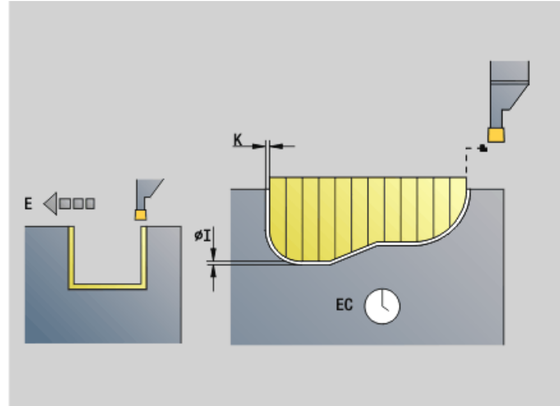
	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

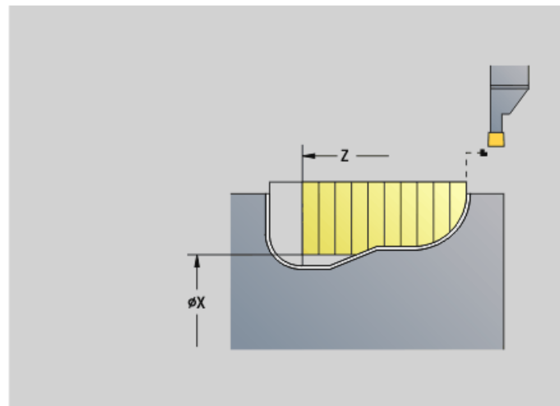


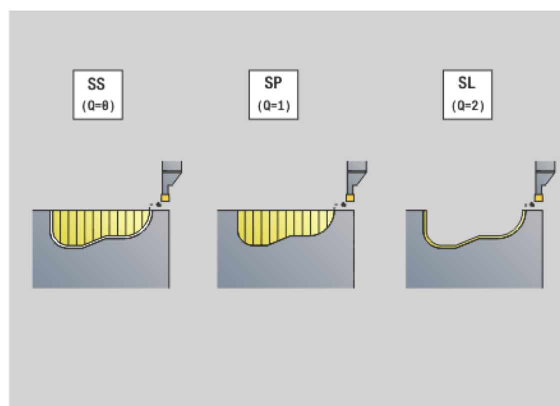


Recessing G860

G860 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call (see "Working with contour-based cycles" on page 262). The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.







Parameters

ID Auxiliary contour—ID number of the contour to be machined

NS Start block number
- Beginning of the contour section, or
- Reference to a G22/G23-Geo recess

NE End block number (end of contour section)
- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.
- NE is inapplicable if the contour is defined by G22/G23-Geo

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

Q Action (default: 0)
- 0: Roughing and finishing
- 1: Only roughing
- 2: Only finishing

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

V Identifier start/end (default: 0) A chamfer/rounding arc is machined:
- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining

E Finishing feed rate (default: active feed rate)

EC Dwell time

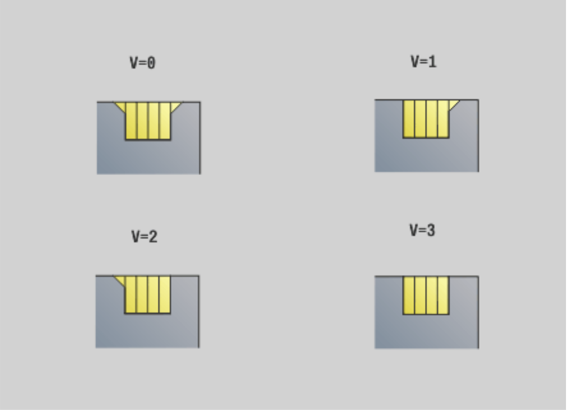
Parameters

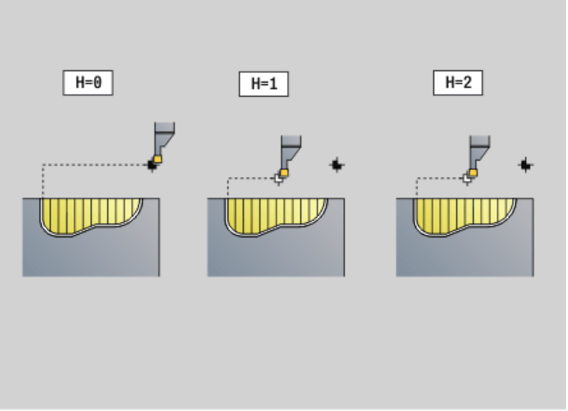
H Type of retraction at cycle end (default: 0)

- 0: Return to starting point
  - Axial recess: First Z, then X direction
  - Radial recess: First X, then Z direction
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

B Recessing width

P Cutting depth by which one cut is fed.





The Control uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Contour cycle repeats can be programmed with G741 before the cycle call.

- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run (where Q=0 or 1)

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account.
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3. Executes first cut (roughing).
4. Returns at rapid traverse and approaches for next pass.
5. Repeats 3 to 4 until the complete area has been machined.
6. If required, repeats 2 to 5 until all areas have been machined.
7. If Q=0: Finish-machines the contour.

Repeat recessing cycle G740/G741

G740 and G741 are programmed before G860 to repeat the recessing contour defined in Cycle G860.

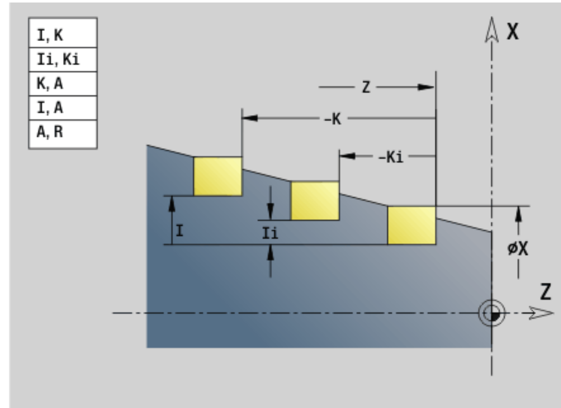
Parameters

X	Starting point X (diameter value) Shifts the starting point of the recessing contour defined by G860 to this coordinate.
Z	Starting point Z. Shifts the starting point of the recessing contour defined by G860 to this coordinate.
I	Distance between the first and last recessing contour (X direction).
K	Distance between the first and last recessing contour (Z direction).
Ii	Distance between the recessing contours (X direction).
Ki	Distance between the recessing contours (Z direction).
Q	Number of recessing contours
A	Angle at which the recessing contours are arranged.
R	Length. Distance between the first and last recessing contour.
Ri	Length. Distance between the recessing contours.

The following parameter combinations are allowed:

- I, K
- Ii, Ki
- I, A
- K, A
- A, R

G740 does not support the parameters A and R.



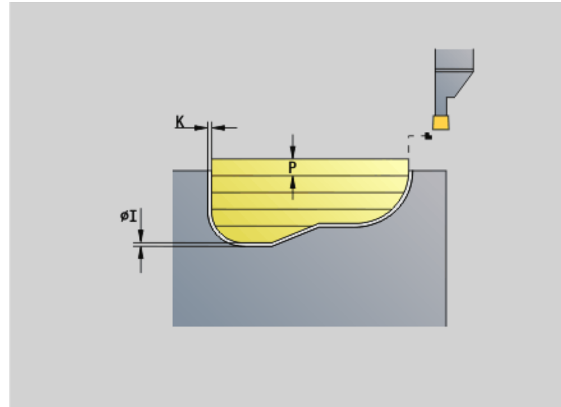
Example: G740, G741

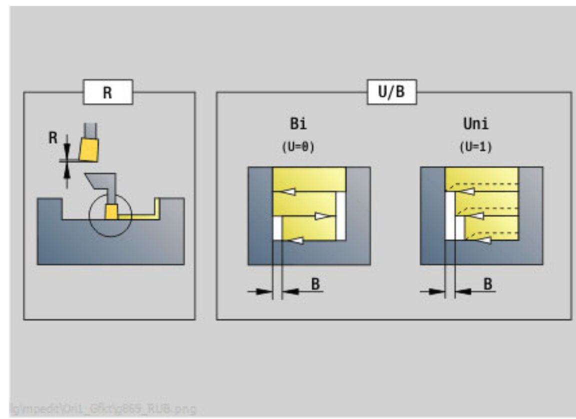
```
. . .
AUXILIARY CONTOUR ID"recess"
N 47 G0 X50 Z0
N  48 G1 Z-5
N  49 G1 X45
N  54 G1 Z-15
N  56 G1 Z-17
MACHINING
N 162 T4
N 163 G96 S150 G95 F0.2 M3
N 165 G0 X120 Z100
N 166 G47 P2
N 167 G741 K-50 Q3 A180
N 168 G860 I0.5 K0.2 E0.15 Q0 H0
N 172 G0 X50 Z0
N 173 G1 X40
N 174 G1 Z-9
N  175 G1 X50
N 169 G80
N 170 G14 Q0
. . .
```

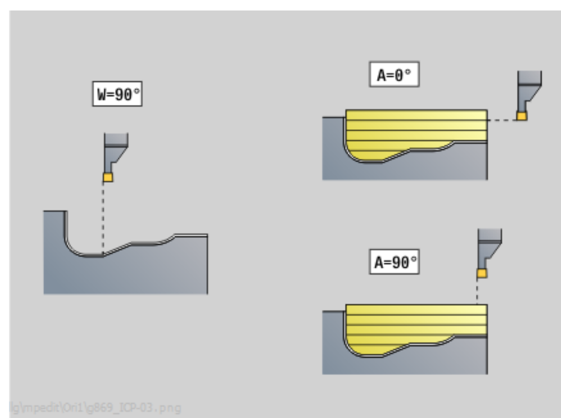
Recess turning cycle G869

G869 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call (see "Working with contour-based cycles" on page 262).

The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements. The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.







Parameters

ID Auxiliary contour—ID number of the contour to be machined

NS Start block number

- Beginning of the contour section, or
- Reference to a G22/G23-Geo recess

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.
- NE is inapplicable if the contour is defined by G22/G23-Geo

P Maximum infeed

R Turning depth compensation for finishing (default: 0)

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

X Cutting limit (diameter value)—(default: no cutting limit)

Z Cutting limit (default: no cutting limit)

A Approach angle (default: opposite to recessing direction)

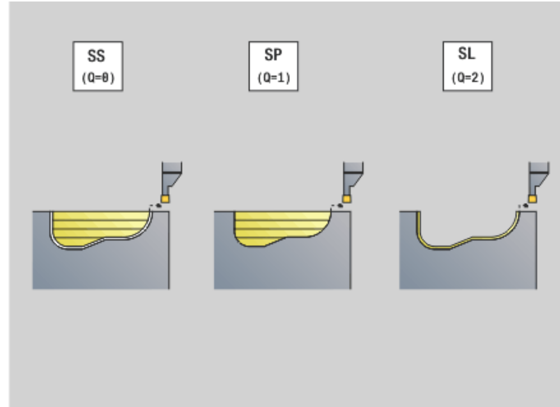
W Departure angle (default: opposite to recessing direction)

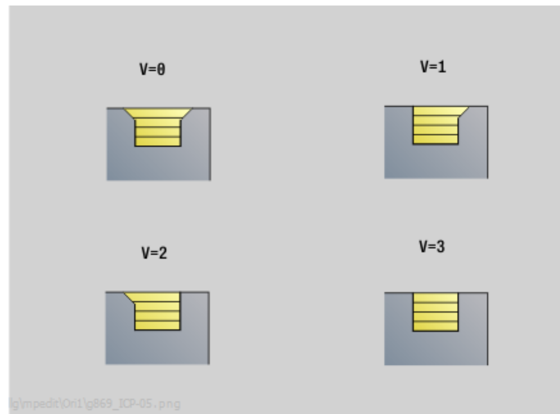
Q Action (default: 0)

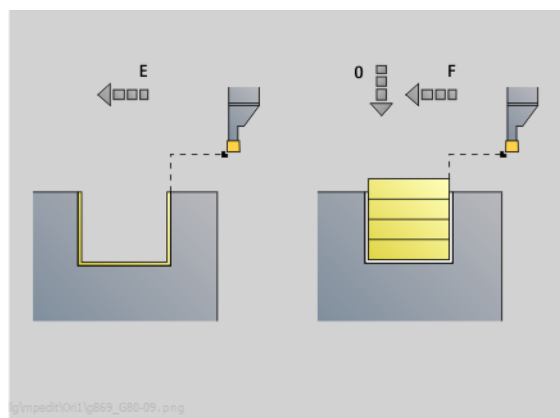
- 0: Roughing and finishing
- 1: Only roughing
- 2: Only finishing

U Unidirectional turning (default: 0)

- 0: The roughing passes are bidirectional.
- 1: The roughing passes are unidirectional (from NS to NE)







Parameters

H Type of retraction at cycle end (default: 0)

- 0: Return to starting point (axial recess: first direction Z, then X; radial recess: first X direction, then Z)
- 1: Positions in front of the finished contour
- 2: Retracts to safety clearance and stops

V Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining

O Recessing feed rate (default: active feed rate)

E Finishing feed rate (default: active feed rate)

B Offset width (default: 0)

XA, ZA Starting point of blank (only effective if no blank was programmed):

- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the
- workpiece blank contour.

The Control uses the tool definition to distinguish between radial and axial recesses.

Program at least one contour reference (e.g.: NS or NS, NE) and P.

Turning depth compensation R: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor. The value is usually determined empirically.

Offset width B: After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the offset width B. Each time the system switches on this side, the path is reduced by B—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius). If required, the Control reduces the programmed offset width. After clearance roughing, the remaining material is removed with a single cut.

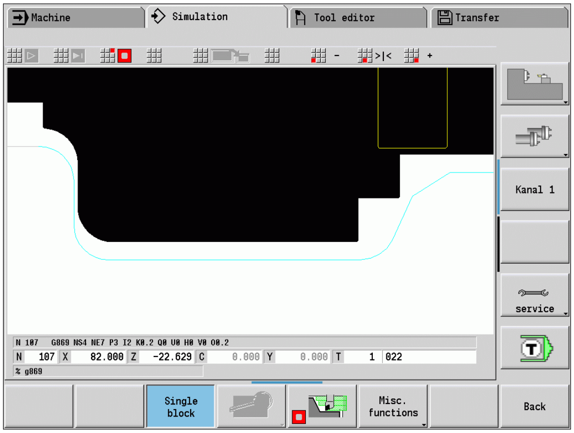
- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run (where Q=0 or 1)

1 Calculates the areas to be machined and the cutting segmentation.
2 Approaches workpiece for first pass from starting point, taking the safety clearance into account.
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3 Executes the first cut (recessing).
4 Machines perpendicularly to recessing direction (turning).
5 Repeats 3 to 4 until the complete area has been machined.
6 If required, repeats 2 to 5 until all areas have been machined.
7 If Q=0: Finish-machines the contour.

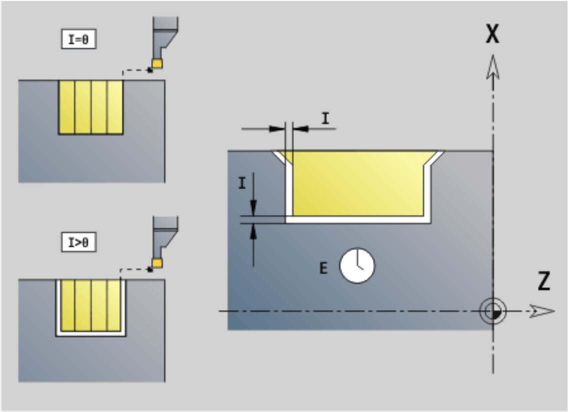
Machining information:

- **Transition from turning to recessing:** Before the transition from turning to recessing, the Control retracts the tool by 0.1 mm. Thus an offset cutting edge is adjusted for the recessing operation, independent of "offset width B."
- **Inside radii and chamfers:** Depending on the recessing width and the radii of rounding arcs, single cuts preventing a "fluid transition" from recessing to turning are executed before the rounding is machined. This prevents damage to the tool.
- **Edges:** Edges are recessed. This prevents residual rings.



Recessing cycle G870

G870 generates a recess defined by G22-Geo. The Control uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.



Parameters

ID Auxiliary contour—ID number of the contour to be machined

NS Block number (reference to G22-Geo)

I Oversize for roughing (default: 0)

- I=0: Recess is made in one work step.
- I>0: The first operation is roughing, the second finishing.

E Dwell time (default: Time for one spindle revolution)

- If I=0: For every recess
- If I>0: Only for finishing

Calculation of cutting segmentation:

Maximum offset = 0.8 * cutting width

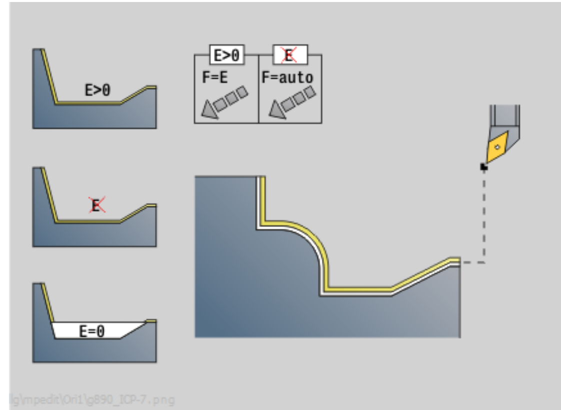
- The **tool radius compensation:** is active.
- An **Oversize** is not taken into account.

Cycle run

1 Calculates the number of cutting passes.
2 Approaches workpiece from starting point for first pass.
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3 Executes the first cut according to I.
4 Returns at rapid traverse and approaches for next pass.
5 If I=0: Dwells for time E
6 Repeats 3 to 4 until the complete recess has been machined.
7 If I>0: Finish machines the contour

Finish contour G890

G890 finishes the defined contour area in one pass. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call (see "Working with contour-based cycles" on page 262). The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.



Parameters

ID Auxiliary contour—ID number of the contour to be machined

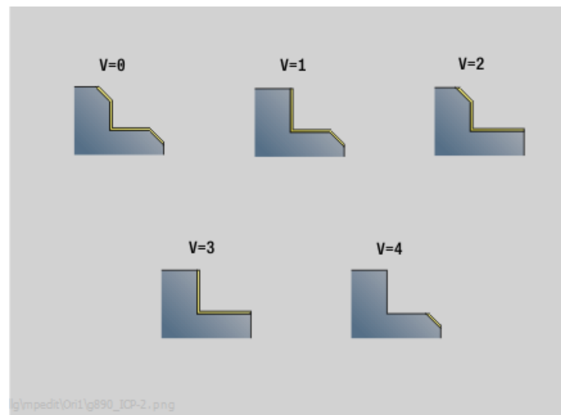
NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

E Plunging behavior

- E=0: Descending contours are not machined
- E>0: Plunging feed rate
- No input: Descending contours are machined at programmed feed rate

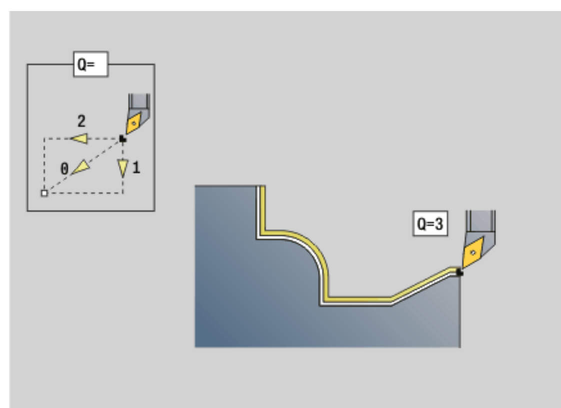


V Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: At beginning and end
- 1: At beginning
- 2: At end
- 3: No machining
- 4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

Q Type of approach (default: 0)

- 0: Automatic selection—the Control checks:
  - Diagonal approach
  - First X, then Z direction
  - Equidistant around the barrier
  - Omission of the first contour elements if the starting position is inaccessible
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: No approach—tool is located near the starting point of the contour area.



Parameters

H Type of retraction (default: 3). Tool backs off at 45° against the machining direction and moves as follows to the position I, K:

- 0: Diagonal
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: Stops at safety clearance
- 4: No retraction motion—tool remains on the end coordinate
- 5: Diagonally to the tool position before the cycle call
- 6: First in X, then in Z to the tool position before the cycle call
- 7: First in Z, then in X to the tool position before the cycle call

X Cutting limit (diameter value)—(default: no cutting limit)

Z Cutting limit (default: no cutting limit)

D Omit elements (default: 1). Use the omit codes listed in the table at right to omit individual elements, or the omit codes listed in the table at the lower right to skip execution of recesses, undercuts and relief turns.

I End point that is approached at the end of the cycle (diameter value)

K End point that is approached at the end of the cycle

O Feed rate reduction for circular elements (default: 0)

- 0: Feed rate reduction is active
- 1: No feed rate reduction

U Cycle type—Required for generating the contour from the G80 parameters. (default: 0)

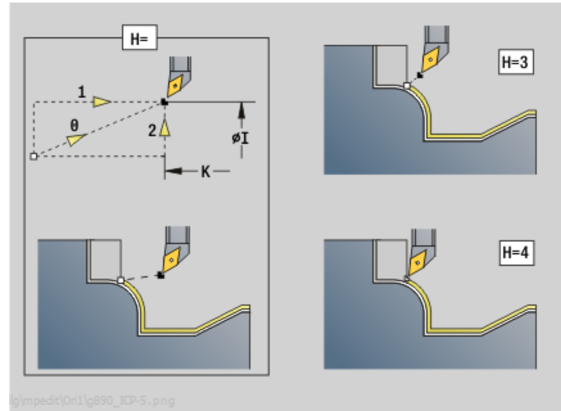
- 0: Standard contour (longitudinal or transverse), recessing contour or ICP contour
- 1: Linear path without/with return
- 2: Circular arc CW, without/with return
- 3: Circular arc CCW, without/with return
- 4: Chamfer without/with return
- 5: Rounding arc without/with return

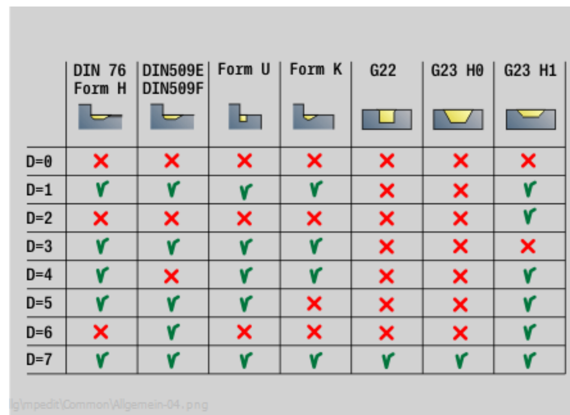
B Tool-tip radius compensation (default: 0)

- 0: Automatic determination
- 1: To the left of the contour
- 2: To the right of the contour

The Control uses the tool definition to distinguish between external and internal machining.

Undercuts are machined if they are programmed and if tool geometry permits.





	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓

Codes for omitting recesses and undercuts		
G call	**Function**	**D code**
G22	Recess for sealing ring	512
G22	Recess for guard ring	1.024
G23 H0	General recess	256
G23 H1	Relief turn	2.048
G25 H4	Undercut type U	32.768
G25 H5	Undercut type E	65.536
G25 H6	Undercut type F	131.072
G25 H7	Undercut type G	262.744
G25 H8	Undercut type H	524.288
G25 H9	Undercut type K	1.048.576
Add the codes if you want to hide several elements.		

Feed rate reduction

- **For chamfers/rounding arcs, the following applies:**
  - Feed rate is programmed with G95-Geo: No automatic feed rate reduction.
  - Feed rate is **not** programmed with G95-Geo: Automatic feed rate reduction. Each chamfer/rounding is therefore machined with at least three revolutions.
  - For chamfers/rounding arcs which, as a result of their size, are machined with at least three revolutions, the feed rate is not reduced automatically.
- **For circular elements, the following applies:**
  - For small circular elements, the feed rate is decreased until every element is machined with at least four spindle revolutions. You can switch this feed rate reduction off with O.
  - The tool radius compensation (TRC) results under certain conditions to a feed rate reduction for circular elements (See "Tool-tip and cutter radius compensation" on page 249.). You can switch this feed rate reduction off with O.

- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Reduces the contour
- **G57/G58 oversizes** are deleted after cycle end.

Measuring cut G809

Cycle G809 performs a cylindrical measuring cut with the length defined in the cycle, moves to the breakpoint for measuring and stops the program. After the program was stopped, you can manually measure the workpiece.

Parameters

X	Starting point X
Z	Starting point Z
R	Measuring cut length
P	Measuring cut oversize
I	Breakpoint Xi for measuring: Incremental distance to starting point of measurement
K	Breakpoint Zi for measuring: Incremental distance to starting point of measurement
ZS	Workpiece blank starting point: Collision-free approach for inside machining
XE	Departing position X
D	Number of an additive compensation to be active during the measuring cut
V	Measuring cut counter: Number of workpieces after which a measurement is performed
Q	Machining direction ■ 0: -Z ■ 1: +Z
EC	Machining location ■ 0: Outside ■ 1: Inside
WE	Directions ■ 0: Simultaneously ■ 1: First X, then Z ■ 2: First Z, then X
O	Approach angle: If an approach angle is entered, the cycle positions the tool over the starting point taking into account the safety clearance and from there plunges at the specified angle to the diameter to be measured.

4.18 Contour definitions in the machining section

Cycle end / Simple contour G80

By programming G80 (with parameters), a turning contour consisting of more than one element can be defined in one NC block. G80 (without parameters) ends a contour definition directly after a cycle.

Parameters

XS Starting point of contour in X (diameter value)
ZS Starting point of contour in Z
XE Contour end point in X (diameter value)
ZE Contour end point in Z
AC Angle of 1st element (range 0°<=AC<90°)
WC Angle of 2nd element (range 0°<=AC<90°)
BS Chamfer/rounding arc at starting point
WS Angle for chamfer at starting point
BE Chamfer/rounding arc at end point
WE Angle for chamfer at end point
RC Radius
IC Chamfer width
KC Chamfer width
JC Execution (see cycle programming)

- 0: Simple contour
- 1: Expanded contour

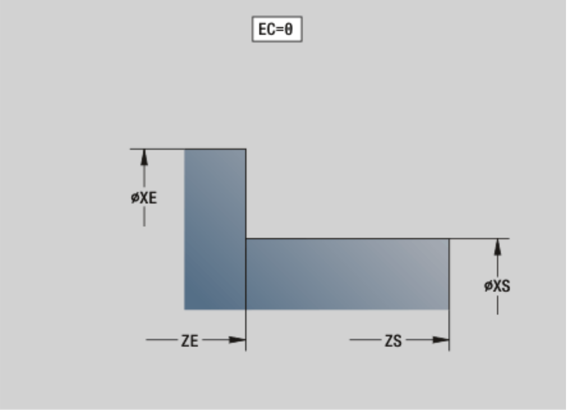
EC Plunging contour

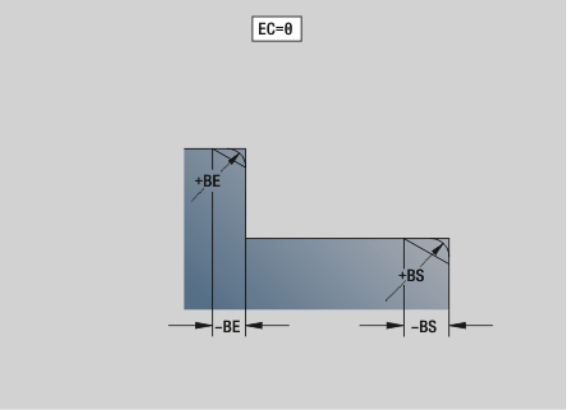
- 0: Rising contour
- 1: Plunging contour

HC Contour direction for finishing:

- 0: Longitudinal
- 1: Transverse

"IC" and "KC" are used in the control to show the chamfer/rounding cycles.



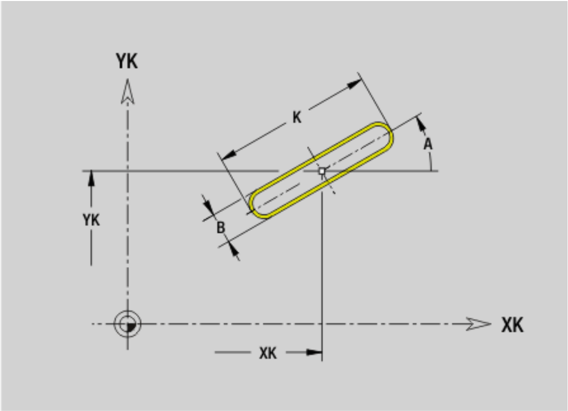


Example: G80

```
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G810 P3
N4 G80 XS60 ZS-2 XE90 ZE-50 BS3 BE-2 RC5
N5 ...
N6 G0 X85 Z2
N7 G810 P5
N8 G0 X0 Z0
N9 G1 X20
N10 G1 Z-40
N11 G80
```

Linear slot on front/rear face G301

G301 defines a linear slot in a contour on the front or rear face. Program this figure in conjunction with G840, G845 or G846.



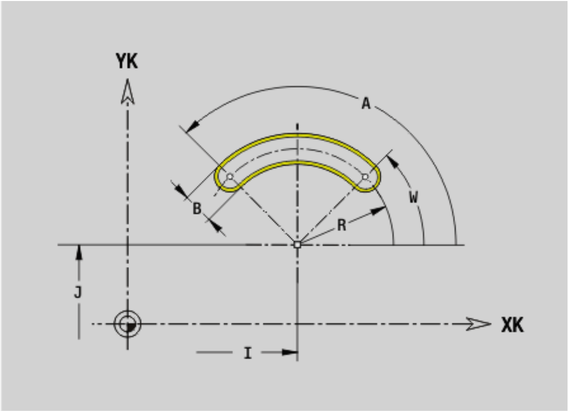
Parameters

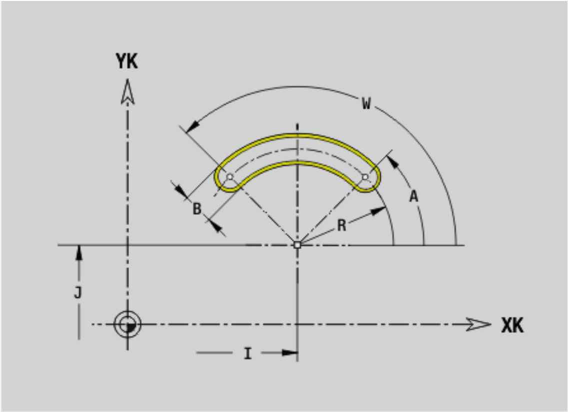
XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
A Angle to XK axis (default: 0°)
K Slot length
B Slot width
P Depth/Height

- P<0: Pocket
- P>0: Island

Circular slot on front/rear face G302/G303

G302/G303 defines a circular slot in a contour on the front face/rear face. Program this figure in conjunction with G840, G845 or G846.





- G302: Circular slot clockwise
- G303: Circular slot counterclockwise

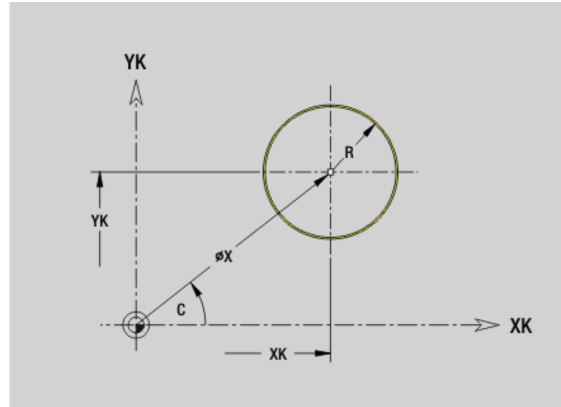
Parameters

I Center of curvature in Cartesian coordinates
J Center of curvature in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
R Curvature radius (reference: center point path of the slot)
A Starting angle; reference: XK axis (default: 0°)
W End angle; reference: XK axis (default: 0°)
B Slot width
P Depth/Height

- P<0: Pocket
- P>0: Island

Full circle on front/rear face G304

G304 defines a full circle in a contour on the front face/rear face. Program this figure in conjunction with G840, G845 or G846.



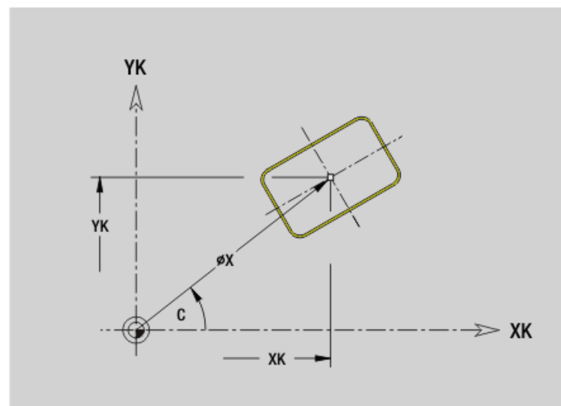
Parameters

XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
R Radius
P Depth/Height

- P<0: Pocket
- P>0: Island

Rectangle on front/rear face G305

G305 defines a rectangle in a contour on the front face/rear face. Program this figure in conjunction with G840, G845 or G846.



Parameters

XK Center in Cartesian coordinates
YK Center in Cartesian coordinates
X Diameter (center point in polar coordinates)
C Angle (center point in polar coordinates)
A Angle to XK axis (default: 0°)
K Length
B (Height) width
R Chamfer/rounding arc (default: 0°)

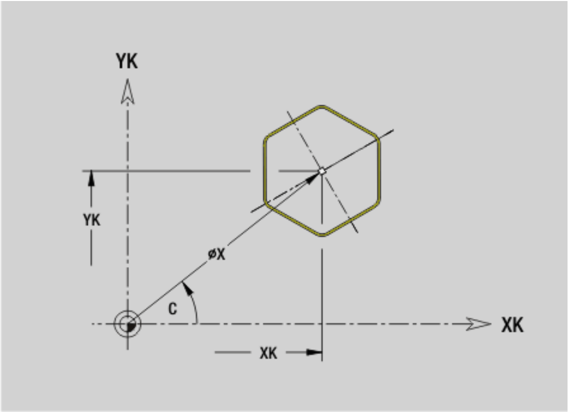
- R>0: Radius of rounding arc
- R<0: Chamfer width

P Depth/Height

- P<0: Pocket
- P>0: Island

Eccentric polygon on front/rear face G307

G307 defines a polygon in a contour on the front face/rear face. Program this figure in conjunction with G840, G845 or G846.

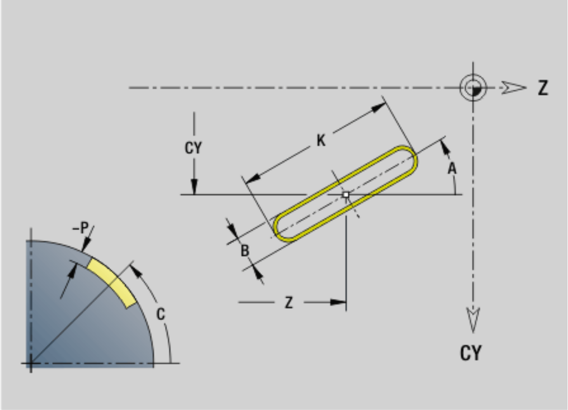


Parameters

- XK Center in Cartesian coordinates
- YK Center in Cartesian coordinates
- X Diameter (center point in polar coordinates)
- C Angle (center point in polar coordinates)
- A Angle of a polygon edge to XK axis (default: 0°)
- Q Number of edges (Q > 2)
- K Edge length
  - K>0: Edge length
  - K<0: Inside diameter of circle
- R Chamfer/rounding arc (default: 0°)
  - R>0: Radius of rounding arc
  - R<0: Chamfer width
- P Depth/Height
  - P<0: Pocket
  - P>0: Island

Linear slot on lateral surface G311

G311 defines a linear slot in a lateral-surface contour. Program this figure in conjunction with G840, G845 or G846.



Parameters

- Z Center (Z position)
- CY Center as linear value; reference: unrolled reference diameter
- C Center (angle)
- A Angle to Z axis (default: 0°)
- K Slot length
- B Slot width
- P Depth of pocket

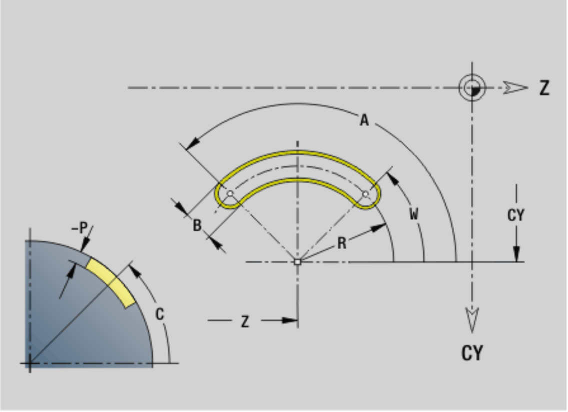
Circular slot on lateral surface G312/G313

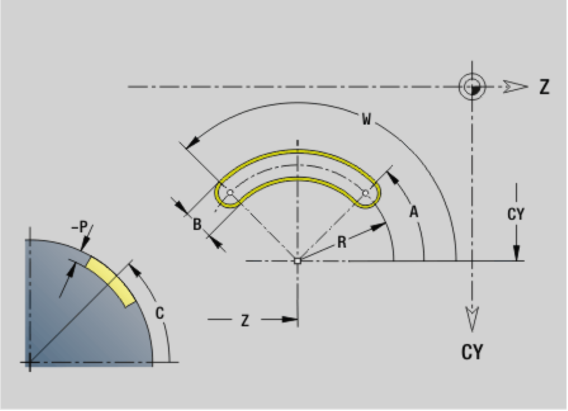
G312/G313 defines a circular slot in a lateral-surface contour Program this figure in conjunction with G840, G845 or G846.

- G312: Circular slot clockwise
- G313: Circular slot counterclockwise

Parameters

Z	Center
CY	Center as linear value; reference: unrolled reference diameter
C	Center (angle)
R	Radius; reference: center point path of the slot
A	Starting angle; reference: Z axis (default: 0°)
W	End angle; reference: Z axis
B	Slot width
P	Depth of pocket



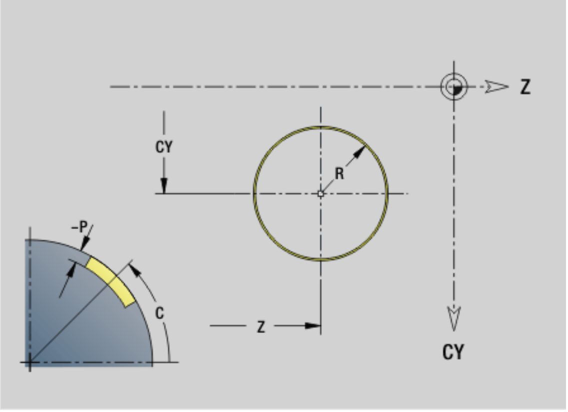


Full circle, lateral surface G314

G314 defines a full circle in a lateral-surface contour. Program this figure in conjunction with G840, G845 or G846.

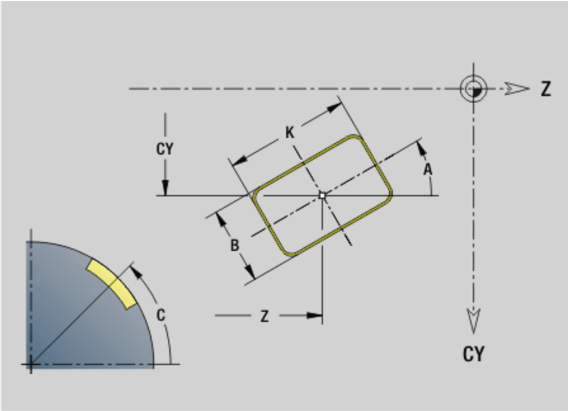
Parameters

Z	Center
CY	Center as linear value; reference: unrolled reference diameter
C	Center (angle)
R	Radius
P	Depth of pocket



Rectangle, lateral surface G315

G315 defines a rectangle in a lateral-surface contour. Program this figure in conjunction with G840, G845 or G846.



Parameters

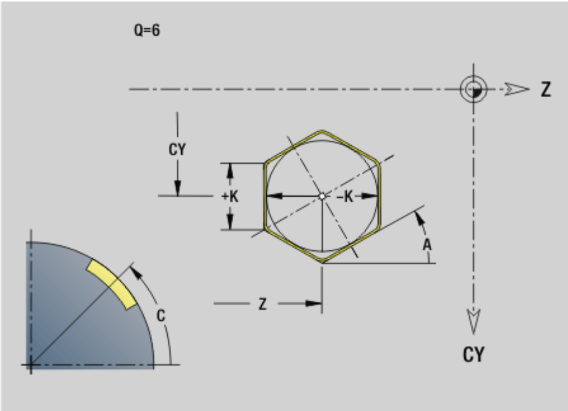
Z Center
CY Center as linear value; reference: unrolled reference diameter
C Center (angle)
A Angle to Z axis (default: 0°)
K Length
B Width
R Chamfer/rounding arc (default: 0°)

- R>0: Radius of rounding arc
- R<0: Chamfer width

P Depth of pocket

Eccentric polygon, lateral surface G317

G317 defines a polygon in a lateral-surface contour. Program this figure in conjunction with G840, G845 or G846.



Parameters

Z Center
CY Center as linear value; reference: unrolled reference diameter
C Center (angle)
Q Number of edges (Q > 2)
A Angle to Z axis (default: 0°)
K Edge length

- K>0: Edge length
- K<0: Inside diameter of circle

R Chamfer/rounding arc (default: 0°)

- R>0: Radius of rounding arc
- R<0: Chamfer width

P Depth of pocket

4.19 Thread cycles

Overview of threading cycles

- G31 machine simple threads, successions of threads and multi-start threads with G24-Geo, G34-Geo or G37-Geo (FINISHED PART). G31 can also machine a threading contour defined directly after the cycle call and concluded by G80: See "Thread cycle G31" on page 293.
- G32 cuts a simple thread in any desired direction and position: See "Simple thread cycle G32" on page 297.
- G33 conducts a single thread cut. The direction of the single thread cut is as desired: See "Thread single path G33" on page 299.
- G35 cuts a simple cylindrical metric ISO thread without run-out: See "Metric ISO thread G35" on page 301.
- cuts a tapered API thread: See "Tapered API thread G352" on page 302.

Handwheel superimposition

If your machine features handwheel superimposition, you can overlap axis movements during thread cutting in a limited area:

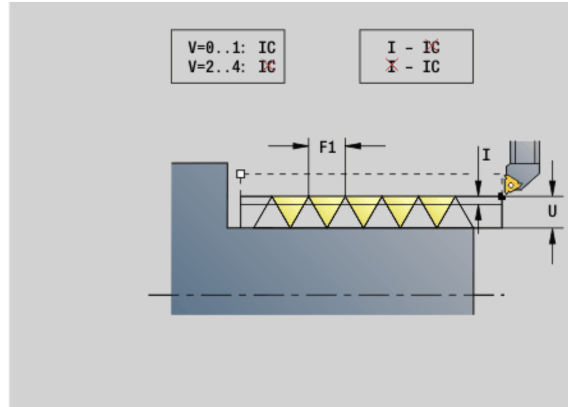
- **X direction**: Maximum programmed thread depth depending on the current cutting depth
- **Z direction**: +/- a fourth of the thread pitch

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

Remember that position changes resulting from handwheel superimposition are no longer effective after the cycle end or the "last cut" function.

Thread cycle G31

G31 machines simple threads, successions of threads and multi-start threads with G24-, G34- or G37-Geo. G31 can also machine a threading contour defined directly after the cycle call and concluded by G80.



Parameters

ID Auxiliary contour—ID number of the contour to be machined

NS Contour start block number (reference to basic element G1-Geo; for successions of threads: block number of the first basic element)

NE Contour end block number (reference to basic element G1-Geo; for successions of threads: block number of the last basic element)

O Identifier start/end (default: 0) A chamfer/rounding arc is machined:

- 0: No machining
- 1: At beginning
- 2: At end
- 3: At beginning and end
- 4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

J Reference direction:

- No input: The reference direction is determined from the first contour element.
- J=0: Longitudinal thread
- J=1: Transverse thread

I Maximum infeed

No input and V=0 (constant chip cross section):
I = 1/3 * F

IC Number of cuts. The infeed is calculated from IC and U. Usable with:

- V=0 (constant chip cross section)
- V=1 (constant infeed)

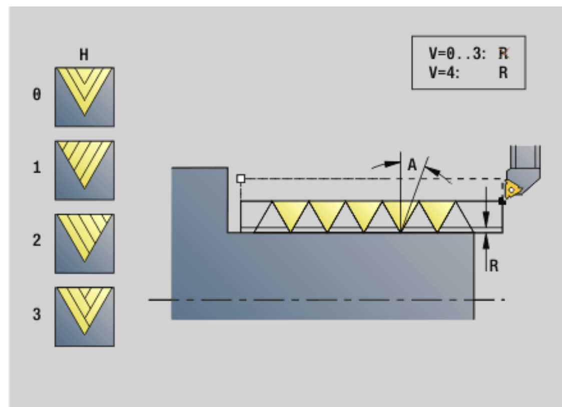
B Run-in length

No input: The run-in length is determined from the contour. If this is not possible, the value is calculated from the kinematic parameters. The thread contour is extended by the value B.

P Overrun length

No input: The run-out length is determined from the contour. If this is not possible, the value is calculated. The thread contour is extended by the value P.

A Approach angle (angle of infeed) (default: 30°)



Example: G31

```
. . .
FINISHED PART
N  2 G0 X16 Z0
N  3 G52 P2 H1
N  4 G95 F0.8
N  5 G1 Z-18
N  6 G25 H7 I1.15 K5.2 R0.8 W30 BF0 BP0
N  7 G37 Q12 F2 P0.8 A30 W30
N  8 G1 X20 BR-1 BF0 BP0
N  9 G1 Z-23.8759 BR0
N  10 G52 G95
N  11 G3 Z-41.6241 I-14.5 BR0
N  12  G1 Z-45
```

Parameters

V Type of infeed (default: 0)

- 0: Constant cross section for all cuts
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts). First infeed = Remainder of the division of thread depth/ cutting depth. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
- 3: Infeed is calculated from the pitch and spindle speed
- 4: Same as MANUALplus 4110

H Type of offset for smoothing the thread flanks (default: 0)

- 0: Without offset
- 1: Offset from the left
- 2: Offset from the right
- 3: Tool is offset alternately from the right and left

R Depth of remaining cuts—only in conjunction with approach type V=4 (same as MANUALplus 4110)

C Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)

BD External/internal thread (no meaning for closed contours)

- 0: External thread
- 1: Internal thread

F Thread pitch

U Thread depth

K Run-out length

- K>0: Run-out
- K<0: Run-in

The length K should be at least the value of the thread depth.

D Number of thread turns for multi-start thread

E Variable pitch (no effect at present)

Q Number of no-load (air) cuts after the last cut (for reducing the cutting pressure in the thread base)—(default: 0)

If a thread has been defined with G24-Geo, G34-Geo or G37-Geo, the parameters F, U, K and D are not relevant.

Example: G31, continued

```
N 13 G1 X30 BR2
N 14 G1 Z-50 BR0
N 15 G2 X36 Z-71 I12 BR5
N 16 G1 X40 Z-80
N 17 G1 Z-99
N 18 G1 Z-100[thread]
N 19 G1 X50
N 20 G1 Z-120
N 21 G1 X0[thread]
N 22 G1 Z0
N 23 G1 X16 BR-1.5
. . .
AUXILIARY CONTOUR ID"thread"
N 24 G0 X20 Z0
N 25 G1 Z-30
N 26 G1 X30 Z-60
N 27 G1 Z-100

MACHINING
N 33 G14 Q0 M108
N 30 T9 G97 S1000 M3
N 34 G47 P2
N 35 G31 NS16 NE17 J0 IC5 B5 P0 V0 H1
BD0 F2 K10
N 36 G0 X110 Z20
N 38 G47 M109
[G80 contours can be inside or outside]
N 43 G31 IC4 B4 P4 A30 V0 H2 C30 BD0 F6
U3 K-10 Q2
N 44 G0 X80 Z0
N 45 G1 Z-20
N 46 G1 X100 Z-40
N 47 G1 Z-60
N 48 G80
[External thread regardless of the value
defined in BD]
N 49 G0 X50 Z-30
```

Run-in length B: The slide requires a run-in distance at the start of thread in order to accelerate to the programmed contouring feed rate before starting the actual thread.

Run-out length P: The slide needs an overtravel at the end of the thread to decelerate again. Remember that the paraxial line P needs overtravel even with an oblique thread run-out

You can calculate the minimum run-in and run-out length with the following equation.

Run-in length: $B = 0.75 * (F*S)^2 / a * 0.66 + 0.15$
Run-out length: $P = 0.75 * (F*S)^2 / a * 0.66 + 0.15$

- F: Thread pitch in mm/revolution
- S: Speed in **revolutions/second**
- a: Acceleration in mm/s² (see axis data)

Determination of external/internal thread:

- G31 with contour reference—closed contour: External or internal thread is defined by the contour. BD has no meaning.
- G31 with contour reference—open contour: External or internal thread is defined by "BD". If "BD" is not programmed, the contour is used to determine whether a thread is external or internal.
- If the thread contour is programmed directly after the cycle, "BD" determines whether the thread is an internal or external thread. If "BD" is not programmed, the algebraic sign of "U" is evaluated (as in the MANUALplus 4110).
  - U>0: Internal thread
  - U<0: External thread

Starting angle C: At the end of the "run-in path B" the spindle is at the "starting angle C" position. Therefore, if the thread is to start exactly at the starting angle, position the tool by the run-in length—or by the run-in length plus a multiple of the thread pitch—in front of the beginning of the thread.

The individual thread cuts are calculated from the thread depth, maximum approach I and type of approach V.

- Cycle stop—the Control retracts the tool from the thread groove and then stops all tool movements. (Lift-off distance: OEM configuration parameter: cfgGlobalProperties-threadliftoff)
- Feed rate override is not effective.

Danger of collision!

An excessive overrun length P might cause a collision. You can check the overrun length during the simulation.

Example: G31, continued

```
N 50 G31 NS16 NE17 O0 IC2 B4 P0 A30 V0
H1 C30 BD1 F2 U1 K10
N 51 G0 Z10 X50
[AUXILIARY CONTOURS can be inside or
outside if they are not closed contours]
N 52 G0 X50 Z-30
N 53 G31 ID"thread" O0 IC2 B4 P0 A30 V0
H1 C30 BD1 F2 U1 K10
N 60 G0 Z10 X50
```

Cycle run

1 Calculates the number of cutting passes.

2 Returns diagonally to the internal starting point at rapid traverse. This point lies in front of the "starting point of thread" by the run-in length B. With H=1 (or 2, 3) the current offset is taken into account for calculating the internal starting point.

The internal starting point is calculated on the basis of the tool tip.

3 Accelerates to feed rate (line B).

4 Executes a thread cut.

5 Decelerates (line P).

6 Retracts to safety clearance, returns at rapid traverse, and approaches for next pass. For multiple threads, the same rate of cut is used for each thread turn, before the next infeed motion is executed.

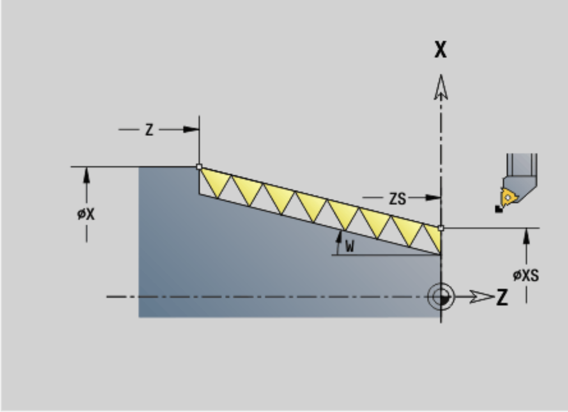
7 Repeats 3 to 6 until the complete thread has been cut.

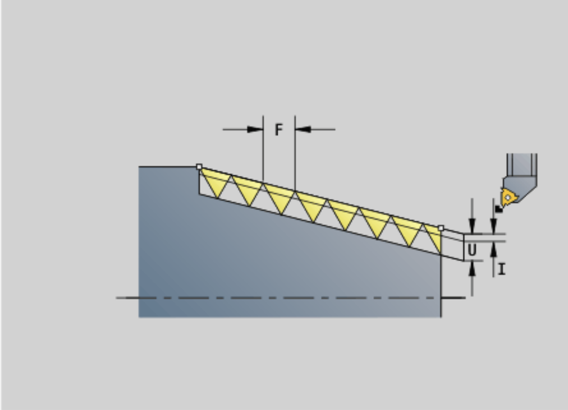
8 Executes air cuts.

9 Returns to starting point.

Simple thread cycle G32

G32 cuts a single thread in any desired direction and position (longitudinal, tapered or transverse thread; internal or external thread).





Parameters

X End point of thread (diameter)
Z End point of thread
XS Starting point for thread (diameter)
ZS Starting point for thread
BD External/internal thread:

- 0: External thread
- 1: Internal thread

F Thread pitch
U Thread depth

No input: The thread depth is calculated automatically:

- External thread (0.6134 * F)
- Internal thread (0.5413 * F)

I Maximum cutting depth
IC Number of cuts. The infeed is calculated from IC and U. Usable with:

- V=0 (constant chip cross section)
- V=1 (constant infeed)

V Type of infeed (default: 0)

- 0: Constant cross section for all cuts
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts). First infeed = Remainder of the division of thread depth/ cutting depth. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8
- 3: Infeed is calculated from the pitch and spindle speed
- 4: Same as MANUALplus 4110

H Type of offset for smoothing the thread flanks (default: 0)

- 0: Without offset
- 1: Offset from the left
- 2: Offset from the right
- 3: Tool is offset alternately from the right and left

K Run-out length at thread end point (default: 0)
W Taper angle (–45° < W < 45°)—(default: 0)

Position of the taper thread with respect to longitudinal or transverse axis:

- W>0: Rising contour (in machining direction)
- W<0: Falling contour

Parameters

C Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)

A Approach angle (angle of infeed) (default: 30°)

R Remainder cuts (default: 0)

- 0: The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
- 1: W/o remaining cutting (without distribution of remaining cuts)

E Variable pitch (no effect at present)

Q Number of no-load (air) cuts after the last cut (for reducing the cutting pressure in the thread base)—(default: 0)

D Number of thread turns for multi-start thread

J Reference direction:

- No input: The reference direction is determined from the first contour element.
- J=0: Longitudinal thread
- J=1: Transverse thread

The cycle calculates the thread from the thread end point, thread depth and the tool position.

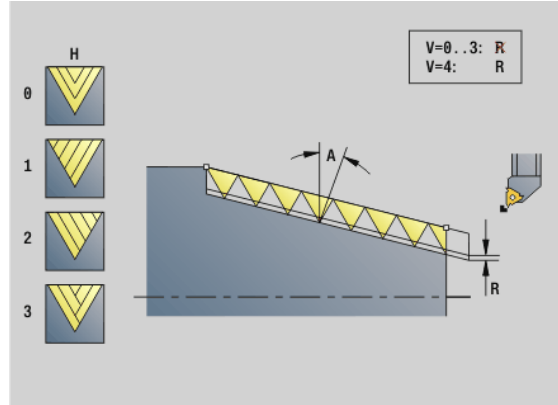
First infeed = Remainder of the division of thread depth/cutting depth.

Transverse thread: Use G31 with contour definition for cutting transverse threads.

- Cycle stop—the Control retracts the tool from the thread groove and then stops all tool movements. (Lift-off distance: OEM configuration parameter: cfgGlobalProperties-threadliftoff)
- Feed rate override is not effective.

Cycle run

1. Calculates the number of cutting passes.
2. Executes a thread cut.
3. Returns at rapid traverse and approaches for next pass.
4. Repeats 2 to 3 until the complete thread has been cut.
5. Executes air cuts.
6. Returns to starting point.



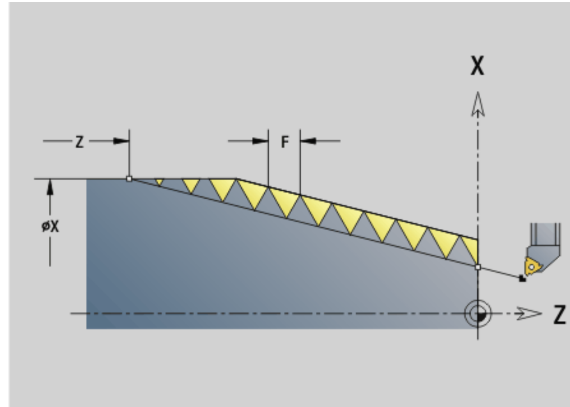
Example: G32

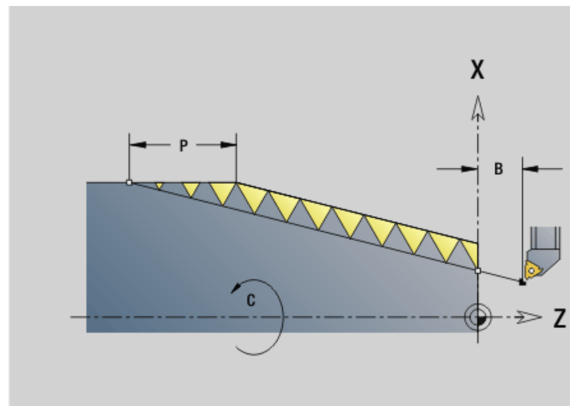
```
. . .
N1 T4 G97 S800 M3
N2 G0 X16 Z4
N3 G32 X16 Z-29 F1.5 [thread]
. . .
```

Thread single path G33

G33 conducts a single thread cut. The direction of the single thread path is as desired (longitudinal, tapered or transverse threads; internal or external threads). You can make successive threads by programming G33 several times in succession.

Position the tool in front of the thread by the run-in length B if the slide must accelerate to the feed rate. And remember the run-out length P **before** the end point of thread if the slide has to be decelerated.





Parameters

- X End point of thread (diameter)
- Z End point of thread
- F Thread pitch
- B Slop. length (run-in length; length of the acceleration path)
- P Overflow length (run-out length; length of the deceleration path)
- C Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)
- H Reference direction for thread pitch (default: 0)
  - 0: Feed rate on the Z axis (for longitudinal and taper threads up to a max. angle of +45°/–45° to the Z axis)
  - 1: Feed rate on the X axis (for transverse and taper threads up to a max. angle of +45°/–45° to the X axis)
  - 3: Contouring feed rate
- E Variable pitch (default: 0)—(no effect at present)
- I Retraction distance in X—retraction path for cycle stop in the thread, incremental value
- K Retraction distance in Z—retraction path for cycle stop in the thread, incremental value

Run-in length B: The slide requires a run-in distance at the start of thread in order to accelerate to the programmed feed rate before starting the actual thread.

Default: cfgAxisProperties/SafetyDist

Run-out length P: The slide needs an overtravel at the end of the thread to decelerate again. Remember that the paraxial line P needs overtravel even with an oblique thread run-out

- P=0: Start of a successive thread
- P>0: End of a successive thread

Starting angle C: At the end of the "run-in path B" the spindle is at the "starting angle C" position.

- Cycle stop—the Control retracts the tool from the thread groove and then stops all tool movements. (Lift-off distance: OEM configuration parameter: cfgGlobalPrperties-threadliftoff)
- Feed rate override is not effective
- Create thread with G95 (feed rate per revolution)

Example: G33

```
. . .
N1 T5 G97 S1100 G95 F0.5 M3
N2 G0 X101.84 Z5
N3 G33 X120 Z-80 F1.5 [thread single path]
N4 G33 X140 Z-122.5 F1.5
N5 G0 X144
. . .
```

Cycle run

1 Accelerates to feed rate (line B).
2 Move to end point of thread—run-out length P
3 Decelerates (line P) remains at the end point of thread.

Activate handwheel during G33

With the G923 function you can activate the handwheel in order to make compensations during a thread cut. In the G923 function you define limits within with traverse with the handwheel is possible.

Parameters

X Max. positive offset: limit in +X
Z Max. positive offset: limit in +Z
U Max. negative offset: limit in –X
W Max. negative offset: limit in –Z
H Reference direction:
- H=0: Longitudinal thread
- H=1: Transversal thread

Q Thread type:
- Q=1: Right-hand thread
- Q=2: Left-hand thread

Metric ISO thread G35

G35 cuts a longitudinal thread (internal or external thread). The thread starts at the current tool position and ends at the end point X, Z.

From the tool position relative to the end point of the thread, Control automatically determines whether an internal or external thread is to be cut.

Parameters

X End point of thread (diameter)

Z End point of thread

F Thread pitch

I Maximum infeed

No input: I is calculated from the thread pitch and the thread depth.

Q Number of no-load (air) cuts after the last cut (for reducing the cutting pressure in the thread base)—(default: 0)

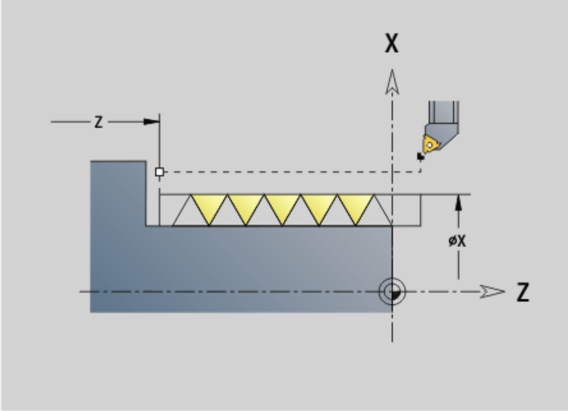
V Type of infeed (default: 0)

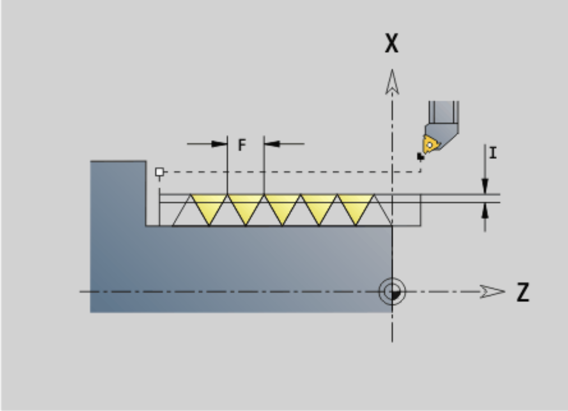
- 0: Constant cross section for all cuts
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts). First infeed = Remainder of the division of thread depth/ cutting depth. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
- 3: Infeed is calculated from the pitch and spindle speed
- 4: Same as MANUALplus 4110

- Cycle stop—the Control retracts the tool from the thread groove and then stops all tool movements. (Lift-off distance: OEM configuration parameter: cfgGlobalPrperties-threadliftoff)
- If you are programming an internal thread, it is advisable to preset the thread pitch F since the diameter of the longitudinal element is not the thread diameter. If you have the Control calculate the thread pitch automatically, slight deviations may occur.

Cycle run

1. Calculates the number of cutting passes.
2. Executes a thread cut.
3. Returns at rapid traverse and approaches for next pass.
4. Repeats 2 to 3 until the complete thread has been cut.
5. Executes air cuts.
6. Returns to starting point.



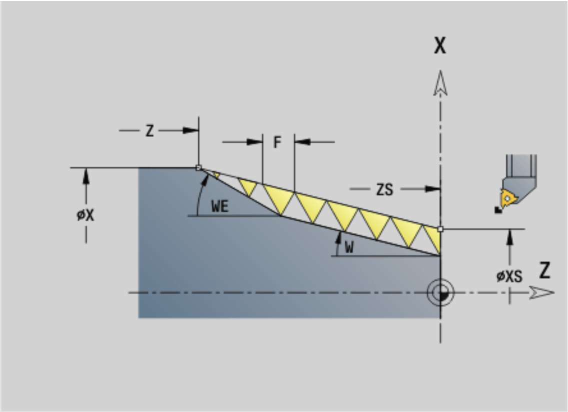


Example: G35

```
%35.nc
[G35]
N1 T5 G97 S1500 M3
N2 G0 X16 Z4
N3 G35 X16 Z-29 F1.5
END
```

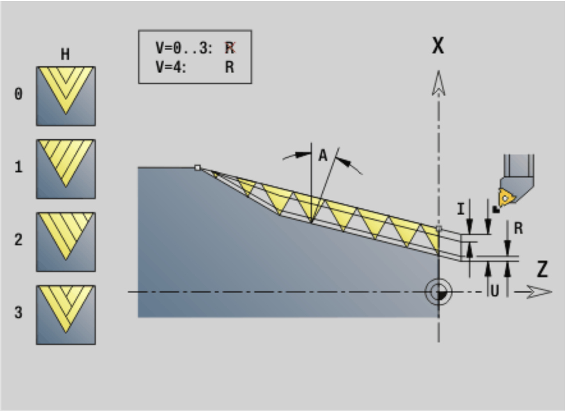
Tapered API thread G352

This cycle cuts a tapered single or multi-start API thread. The depth of thread decreases at the overrun at the end of thread.



Parameters

X	End point of thread (diameter)
Z	End point of thread
XS	Starting point for thread (diameter)
ZS	Starting point for thread
F	Thread pitch
U	Thread depth ■ U>0: Internal thread ■ U<=0: External thread (lateral surface or front face) ■ U= +999 or –999: Thread depth is calculated
I	Maximum approach (infeed) (default: I is calculated from the thread pitch and the thread depth)
V	Type of infeed (default: 0) ■ 0: Constant cross section for all cuts ■ 1: Constant infeed ■ 2: W/ remaining cutting (with distribution of remaining cuts). First infeed = Remainder of the division of thread depth/ cutting depth. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8 ■ 3: Infeed is calculated from the pitch and spindle speed ■ 4: Same as MANUALplus 4110
H	Type of offset for smoothing the thread flanks (default: 0) ■ 0: Without offset ■ 1: Offset from the left ■ 2: Offset from the right ■ 3: Tool is offset alternately from the right and left
A	Approach (infeed) angle (range: –60° < A < 60°; default 30°) ■ A>0: Infeed on right thread flank ■ A<0: Infeed on left thread flank
R	Depth of remaining cuts—only in conjunction with approach type V=4 (same as MANUALplus 4110)
W	Cone (taper) angle (range: –45° < W < 45°, default: 0°)
WE	Run-out angle (range: 0° < WE < 90°, default: 12°)
D	Threads per unit (number of thread turns) for multi-start thread
Q	Number of no-load (air) cuts after the last cut (for reducing the cutting pressure in the thread base)—(default: 0)
C	Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)



Example: G352

```
%352.nc
[G352]
N1 T5 G97 S1500 M3
N2 G0 X13 Z4
N3 G352 X16 Z-28 XS13 ZS0 F1.5 U-999
WE12
END
```

Internal or external threads: See algebraic sign of "U."

Number of cutting passes: The first cut is performed at the cutting depth defined for "I" and is reduced with each cut until the tool reaches the "remaining cutting depth R."

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depending on the current cutting depth—without exceeding the starting and end points of the thread.
- **Z direction:** Maximal 1 thread groove—without exceeding the starting and end points of the thread.

Definition of **taper angle**:

- XS/ZS, X/Z
- XS/ZS, Z, W
- ZS, X/Z, W

- Cycle stop—the Control retracts the tool from the thread groove and then stops all tool movements. (Lift-off distance: OEM configuration parameter: cfgGlobalPrperties-threadliftoff)
- If you are programming an internal thread, it is advisable to preset the thread pitch F since the diameter of the longitudinal element is not the thread diameter. If you have Control calculate the thread pitch automatically, slight deviations may occur.

Cycle run

1. Calculates the number of cutting passes.
2. Executes a thread cut.
3. Returns at rapid traverse and approaches for next pass.
4. Repeats 2 to 3 until the complete thread has been cut.
5. Executes air cuts.
6. Returns to starting point.

Metric ISO thread G38

Cycle G38 creates a cylindrical thread whose form does not correspond to the tool form. Use a recessing or button tool for machining.

Describe the contour of the thread turn as auxiliary contour. The position of the auxiliary contour must correspond to the start position of the thread cuts. You can select the entire auxiliary contour or just segments in the cycle.

Parameters

ID Name of the auxiliary contour
NS Start block of the contour to be machined
NE End block of the contour to be machined
Q Thread depth

- 0: Roughing: The contour is roughed out line by line at maximum infeed **I** and **K**. A programmed oversize (G58 or G57) is taken into account.
- 1: Roughing: The turn of the thread is created in individual cuts along the contour. Define the distances between the individual thread cuts on the contour with **I** and **K**.

X End point of thread X
Z End point of thread Z
F Thread pitch
I Maximum infeed

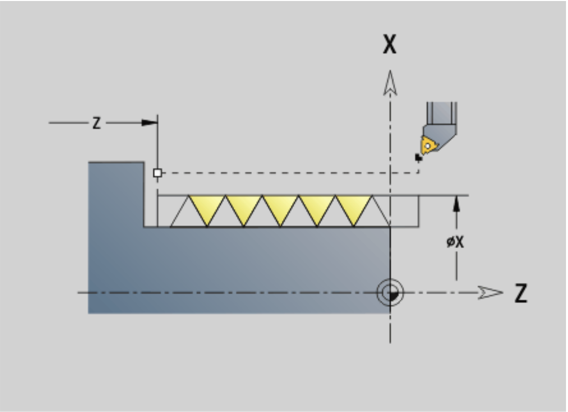
- If Q=0: Plunging depth
- If Q=1: Distance between the finishing cuts as arc length

K Maximum infeed

- If Q=0: Offset width
- If Q=1: Distance between the finishing cuts on straight line

J Run-out length
C Starting angle
O Type of infeed

- 0: Rapid traverse
- 1: Feed rate



Example: G38

```
%352.nc
[G38]
N1 T5 G97 S1500 M3
N2 G0 X43 Z4
N3 G38 ID"123" NS3 NE5 X40 Z-30 F1.5 I0.8
K0.5 J3 C0
END
```

4.20 Parting cycle

Cut-off cycle G859

Cycle G859 parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter. At the end of cycle, the tool retracts and returns to the starting point.

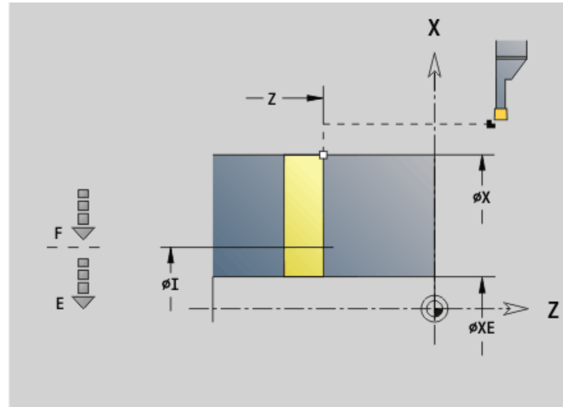
You can define a feed rate reduction, which becomes effective as soon as the position I is reached.

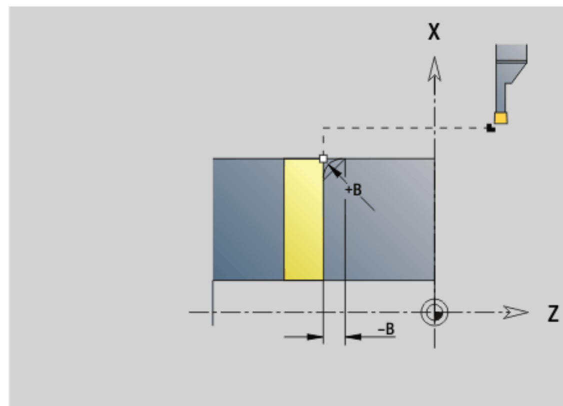
Parameters

X Cut-off (parting) diameter
Z Cut-off (parting) position
I Diameter for feed rate reduction
- I is defined: The control switches to feed rate E after this position
- I is not defined: No feed rate reduction

XE Inside diameter (pipe)
E Reduced feed rate
B Chamfer/rounding
- B>0: Radius of rounding
- B<0: Chamfer width

D Speed limitation: maximum speed during parting





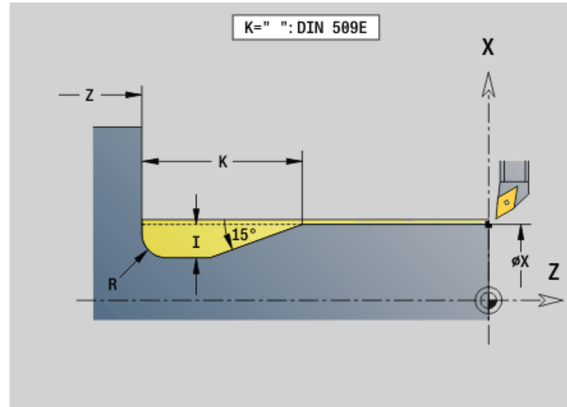
Example: G859

```
%859.nc
[G859]
N1 T3 G95 F0.23 G96 S248 M3
N2 G0 X60 Z-28
N3 G859 X50 Z-30 I10 XE8 E0.11 B1
END
```

4.21 Undercut cycles

Undercut cycle G85

With the function G85, you can machine undercuts according to DIN 509 E, DIN 509 F and DIN 76 (thread undercut).



Parameters

X Target point (diameter)

Z Target point

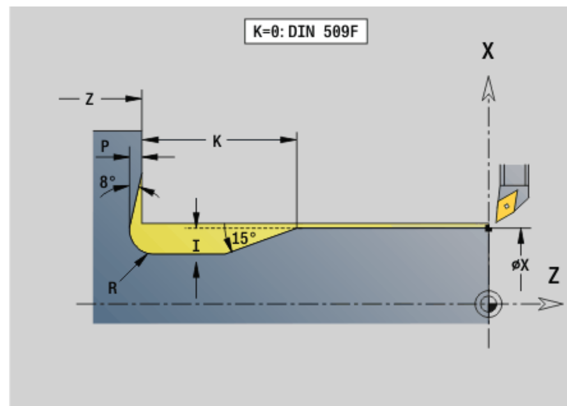
I Depth (radius)

- DIN 509 E, F: Finishing oversize (default: 0)
- DIN 76: Undercut depth

K Undercut width and **type of undercut**

- K No input: DIN 509 E
- K=0: DIN 509 F
- K>0: Undercut width for DIN 76

E Reduced feed for machining the undercut (default: active feed rate)

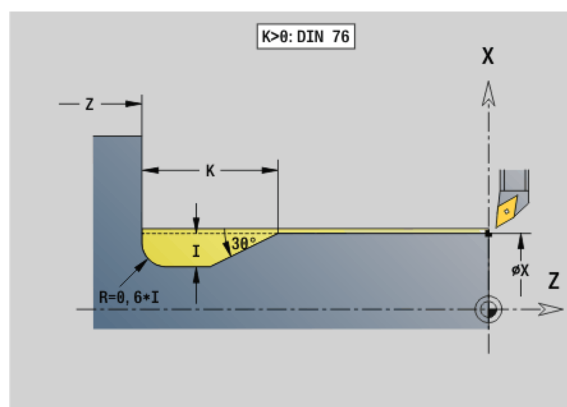


G85 machines the adjoining cylinder if you position the tool to diameter X "in front of" the cylinder.

The undercut rounding arcs are executed with the radius 0.6 * I.

Parameters for undercut DIN 509 E			
Diameter	**I**	**K**	**R**
<= 18	0.25	2	0.6
> 18 – 80	0.35	2.5	0.6
> 80	0.45	4	1

Parameters for undercut DIN 509 E				
Diameter	**I**	**K**	**R**	**P**
<= 18	0.25	2	0.6	0.1
> 18 – 80	0.35	2.5	0.6	0.2
> 80	0.45	4	1	0.3



- I = undercut depth
- K = undercut width
- R = undercut radius
- P = face depth
- **Undercut angle** for undercuts according to DIN 509 E and F: 15°
- **Transverse angle** for an undercut according to DIN 509 F: 8°

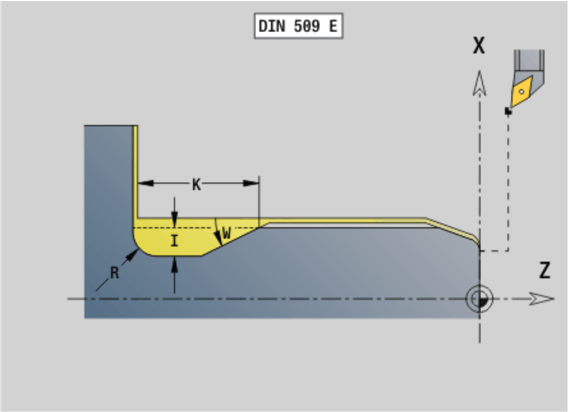
- The **tool radius compensation** is not active.
- **Oversizes** are not taken into account.

Example: G85

```
. . .
N1 T21 G95 F0.23 G96 S248 M3
N2 G0 X62 Z2
N3 G85 X60 Z-30 I0.3
N4 G1 X80
N5 G85 X80 Z-40 K0
N6 G1 X100
N7 G85 X100 Z-60 I1.2 K6 E0.11
N8 G1 X110
. . .
```

Undercut according to DIN 509 E with cylinder machining G851

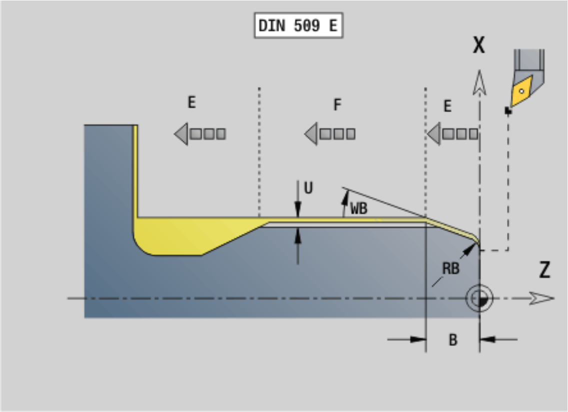
G851 machines the adjoining cylinder, the undercut, and finishes with the plane surface. It also machines a cylinder start chamfer when you enter at least one of the parameters **Cut-in length (1st cut length)** or **Cut-in radius (1st cut radius)**.



Parameters

I	Undercut depth (default: value from standard table)
K	Undercut length (default: value from standard table)
W	Undercut angle (default: value from standard table)
R	Undercut radius (default: value from standard table)
B	Cut-in length (1st cut length)—no input: No chamfer machined at start of cylinder
RB	Cut-in radius (1st cut radius)—no input: 1st cut radius is not machined
WB	1st cut angle (default: 45°)
E	Reduced feed for machining the undercut (default: active feed rate)
H	Type of departure (default: 0): ■ 0: Tool returns to the starting point ■ 1: Tool remains at the end of the plane surface
U	Grinding oversize for the area of the cylinder (default: 0)

The Control calculates unentered parameters from the diameter of the cylinder in the standard table (see "Undercut cycle G85" on page 306).



Blocks following the cycle call

```
N.. G851 I.. K.. W..  /Cycle call
N.. G0 X.. Z..        /Corner point of cylinder start chamfer
N.. G1 Z..            /Undercut corner
N.. G1 X..            /End point of plane surface
N.. G80  /End of contour definition
```

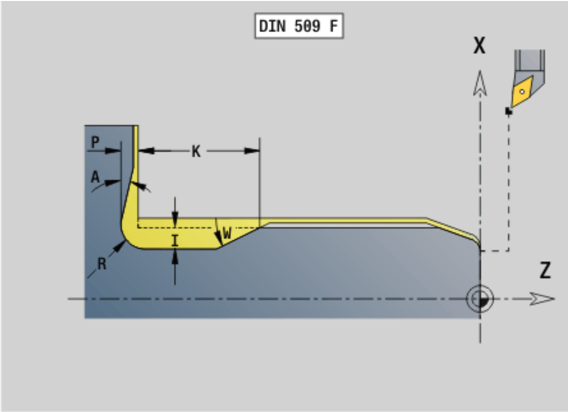
- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

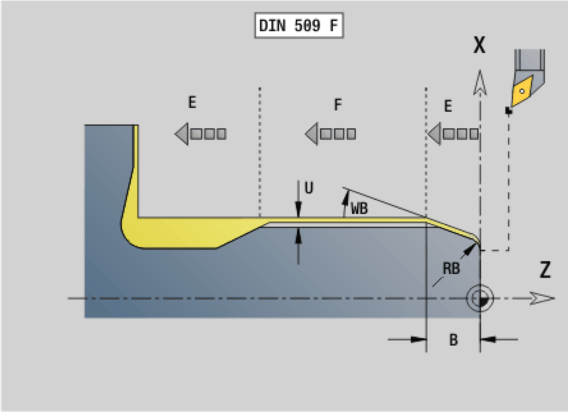
Example: G851

```
%851.nc
[G851]
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G851 I3 K15 W30 R2 B5 RB2 WB30 E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut according to DIN 509 F with cylinder machining G852

G852 machines the adjoining cylinder, the undercut, and finishes with the plane surface. It also machines a cylinder start chamfer when you enter at least one of the parameters **Cut-in length (1st cut length)** or **Cut-in radius (1st cut radius)**.





Parameters

I	Undercut depth (default: value from standard table)
K	Undercut length (default: value from standard table)
W	Undercut angle (default: value from standard table)
R	Undercut radius (default: value from standard table)
P	Face depth (default: value from standard table)
A	Face angle (default: value from standard table)
B	Cut-in length (1st cut length)—no input: No chamfer machined at start of cylinder
RB	Cut-in radius (1st cut radius)—no input: 1st cut radius is not machined
WB	1st cut angle (default: 45°)
E	Reduced feed for machining the undercut (default: active feed rate)
H	Type of departure (default: 0): ■ 0: Tool returns to the starting point ■ 1: Tool remains at the end of the plane surface
U	Grinding oversize for the area of the cylinder (default: 0)

The Control calculates unentered parameters automatically from the diameter in the standard table (see "Undercut cycle G85" on page 306).

Blocks following the cycle call

```
N.. G852 I.. K.. W..  /Cycle call
N.. G0 X.. Z..        /Corner point of cylinder start chamfer
N.. G1 Z..            /Undercut corner
N.. G1 X..            /End point of plane surface
N.. G80  /End of contour definition
```

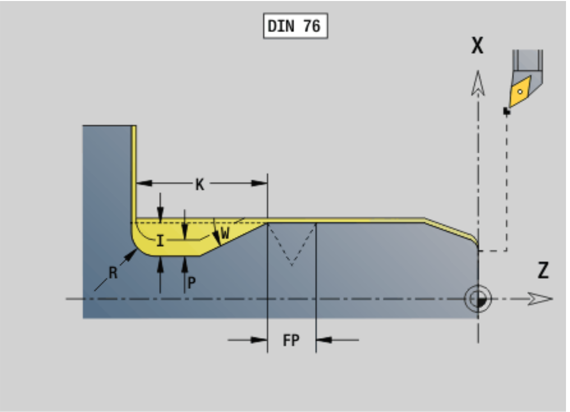
- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

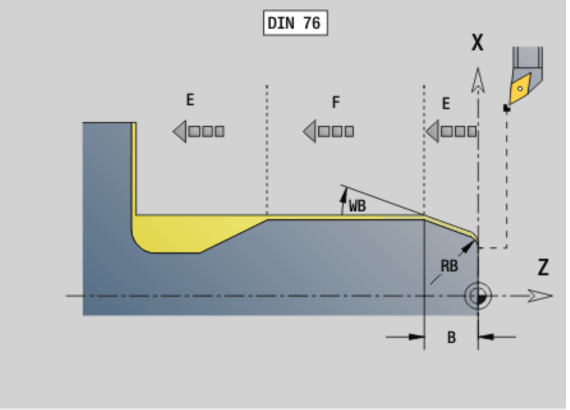
Example: G852

```
%852.nc
[G852]
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G852 I3 K15 W30 R2 P0.2 A8 B5 RB2 WB30
E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut according to DIN 76 with cylinder machining G853

G853 machines the adjoining cylinder, the undercut, and finishes with the plane surface. It also machines a cylinder start chamfer when you enter at least one of the parameters **`Cut-in length (1st cut length)`** or **`Cut-in radius (1st cut radius)`**.





Parameters

FP	Thread pitch
I	Undercut depth (default: value from standard table)
K	Undercut length (default: value from standard table)
W	Undercut angle (default: value from standard table)
R	Undercut radius (default: value from standard table)
P	Oversize: ■ P is not defined: The undercut is machined in one pass ■ P is defined: Division into pre-turning and finish-turning – P = longitudinal oversize; the transverse oversize is preset to 0.1 mm
B	Cut-in length (1st cut length)—no input: No chamfer machined at start of cylinder
RB	Cut-in radius (1st cut radius)—no input: 1st cut radius is not machined
WB	1st cut angle (default: 45°)
E	Reduced feed for machining the undercut (default: active feed rate)
H	Type of departure (default: 0): ■ 0: Tool returns to the starting point ■ 1: Tool remains at the end of the plane surface

Parameters that are not programmed are automatically calculated by the Control from the standard table.

- FP from the diameter
- I, K, W, and R from FP (thread pitch)

Blocks following the cycle call

```
N.. G853 FP.. I.. K.. W..  /Cycle call
N.. G0 X.. Z..             /Corner point of cylinder start chamfer
N.. G1 Z..                 /Undercut corner
N.. G1 X..                 /End point of plane surface
N.. G80  /End of contour definition
```

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

Example: G853

```
%853.nc
[G853]
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G853 FP1.5 I47 K15 W30 R2 P1 B5 RB2
WB30 E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut type U G856

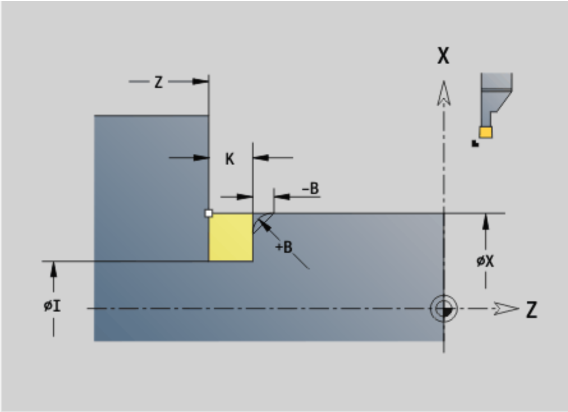
G856 machines an undercut and finishes the adjoining plane surface. A chamfer or rounding (optional) can be machined.

Tool position at the end of the cycle: Cycle starting point

Parameters

I Undercut depth (default: value from standard table)
K Undercut length (default: value from standard table)
B Chamfer/rounding:

- B>0: Radius of rounding
- B<0: Chamfer width



Blocks following the cycle call

N.. G856 I.. K.. /Cycle call
N.. G0 X.. Z.. /Undercut corner
N.. G1 X.. /End point of plane surface
N.. G80 /End of contour definition

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.
- If the cutting width of the tool is not defined, the control assumes that the tool's cutting width equals K.

Example: G856

```
%856.nc
[G856]
N1 T3 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G856 I47 K7 B1
N4 G0 X50 Z-30
N5 G1 X60
N6 G80
END
```

Undercut type H G857

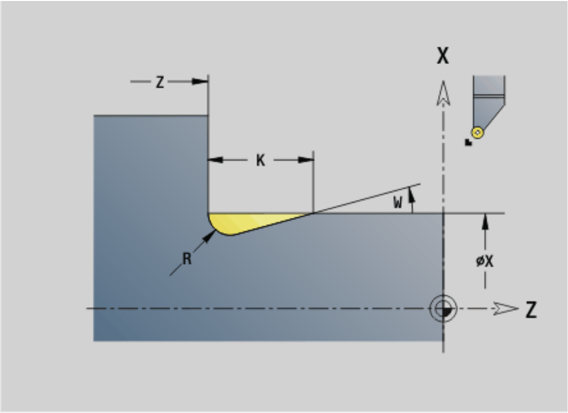
G857 machines an undercut. The end point is determined from the plunge angle in accordance with **Undercut type H**.

Tool position at the end of the cycle: Cycle starting point

Parameters

X	Corner point of contour (diameter)
Z	Corner point of contour
K	Undercut length
R	Radius—no input: No circular element (tool radius = undercut radius)
W	Plunging angle—no input: W is calculated from K and R

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.



Example: G857

```
%857.nc
[G857]
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G857 X50 Z-30 K7 R2 W30
END
```

Undercut type K G858

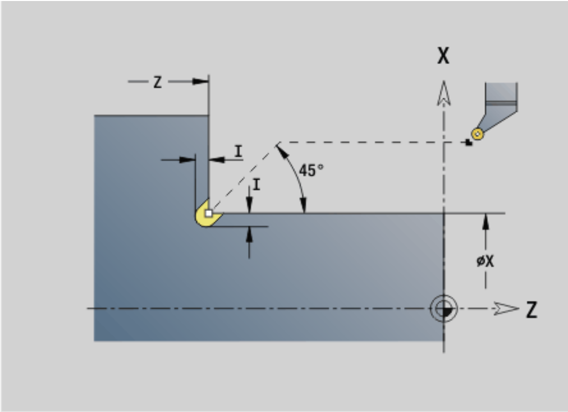
G858 machines an undercut. This cycle performs only one cut at an angle of 45°. The resulting contour geometry therefore depends on the tool that is used.

Tool position at the end of the cycle: Cycle starting point

Parameters

X Corner point of contour (diameter)
Z Corner point of contour
I Undercut depth

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.



Example: G858

```
%858.nc
[G858]
N1 T9 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G858 X50 Z-30 I0.5
END
```

4.22 Drilling cycles

Overview of drilling and boring cycles and contour reference

The drilling and boring cycles can be used with driven or stationary tools.

Drilling and boring cycles:

- G71 Simple drilling: Page 315
- G72 Boring/countersinking (only with contour reference (ID, NS): Page 317
- G73 Tapping (not with G743–G746): Page 324
- G74 Deep-hole drilling: Page 321
- G36 Tapping—single path (direct position input): Page 320
- G799 Thread milling (direct position input): Page 328

Pattern definitions:

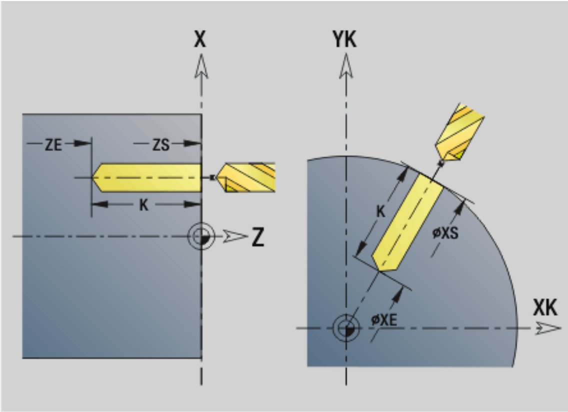
- G743 Linear pattern on face for drilling and milling cycles: Page 324
- G744 Linear pattern on lateral surface for drilling and milling cycles: Page 326
- G745 Circular pattern on face for drilling and milling cycles: Page 325
- G746 Circular pattern on lateral surface for drilling and milling cycles: Page 327

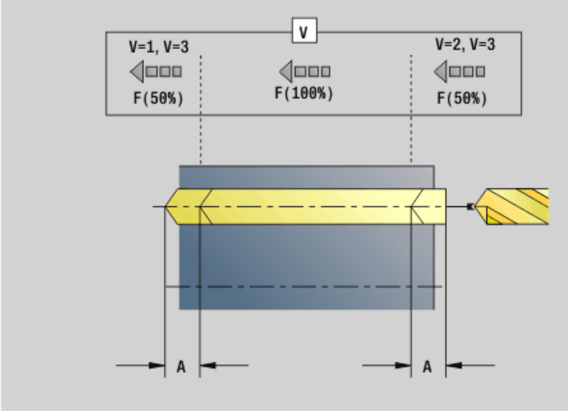
Possibilities of defining a contour reference:

- Path definition directly in the cycle.
- Reference to a hole or pattern definition in the contour section (ID, NS) for machining on the front face or lateral surface.
- Centric hole in the turning contour (G49): Page 212
- Pattern definition in the block before the cycle call (G743 - G746)

Drilling cycle G71

G71 is used for axial and radial bore holes using driven or stationary tools.





Parameters

ID Drilling contour—Name of the hole definition

NS Block number of contour

- Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)
- No input: Single hole without contour description

XS Starting point of radial hole (diameter value)

ZS Starting point of axial hole

XE End point radial hole (diameter value)

ZE End point of axial hole

K Boring depth (hole depth) (alternative to XE/ZE)

A Drilling lengths (default: 0)

V Bore (through-drilling) variant (feed rate reduction 50%)—(default: 0)

- 0: No feed rate reduction
- 1: Feed reduction for through-drilling
- 2: Feed reduction for pre-drilling
- 3: Feed reduction for pre-drilling and through-drilling

RB Retraction plane (radial holes, holes in the YZ plane: diameter)—(default: retract to starting position or to safety clearance)

E Period of dwell for chip breaking at end of hole (in seconds)—(default: 0)

D Retraction type (default: 0)

- 0: Rapid traverse
- 1: Feed rate

BS Start element no. (number of the first hole to be machined in a pattern)

BE End element no. (number of the last hole to be machined in a pattern)

H (Spindle) Brake off (default: 0)

- 0: Spindle brake on
- 1: Spindle brake off

- Single hole without contour description: Program XS or ZS as alternative.
- Hole with contour description: Do not program XS, ZS.
- Hole pattern: NS refers to the hole contour, and not the definition of the pattern.

Example: G71

```
. . .
N1 T5 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-25 A5 V2 [drilling]
. . .
```

Parameter combinations for single holes without contour description	
XS, XE	ZS, ZE
XS, K	ZS, K
XE, K	ZE, K

Feed rate reduction:

- Indexable insert drill and twist drill with 180° drilling angle
  - A feed rate reduction is only effective if the parameter "Drilling length A" has been defined.
- Other drills
  - Beginning of hole: Feed rate reduction as programmed in V
  - End of hole: Reduction as of hole end point—length of first cut—safety clearance
- Length of first cut = tool tip
- Safety clearance: See user parameter or G47, G147

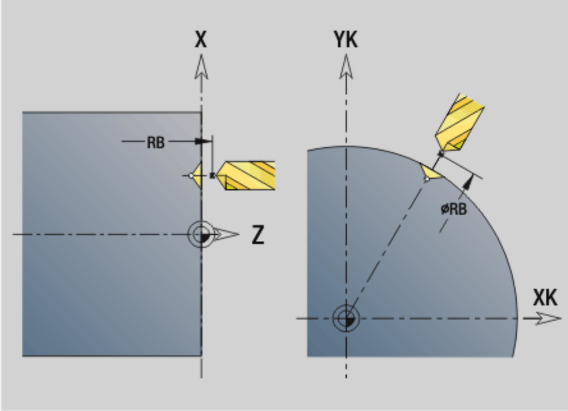
Cycle run

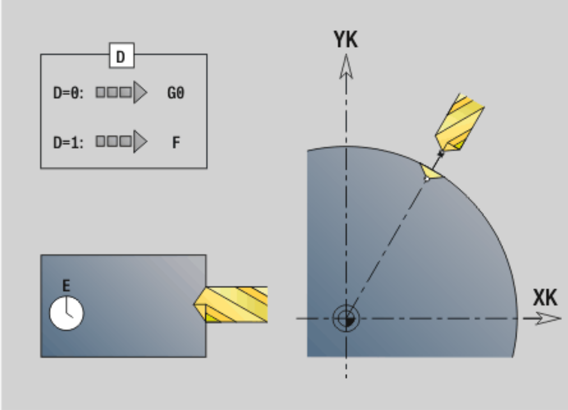
1. - **Hole without contour definition:** Tool is located at the starting point (safety distance from the bore hole).
   - **Hole with contour description:** Tool moves at rapid traverse to the starting point:
     - RB not programmed: Moves up to the safety clearance
     - RB programmed: Moves to the position RB and then to the safety clearance
2. Spot drilling. Feed rate reduction depending on V
3. Drilling at feed rate.
4. Through drilling. Feed rate reduction depending on V
5. Retract at rapid traverse or feed rate, depending on D.
6. Return position:
   - RB not programmed: Retraction to the starting point
   - RB programmed: Retraction to the position RB

Boring, countersinking G72

G72 is used for holes with contour definition (individual hole or hole pattern). Use G72 for the following axial and radial drilling functions using driven or stationary tools:

- Boring
- Countersinking
- Reaming
- NC drilling
- Centering





Parameters

ID Drilling contour—Name of the hole definition

NS Block number of contour. Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)

E Period of dwell for chip breaking at end of hole (in seconds)—(default: 0)

D Retraction type (default: 0)

- 0: Rapid traverse
- 1: Feed rate

BS Start element no. (number of the first hole to be machined in a pattern)

BE End element no. (number of the last hole to be machined in a pattern)

H (Spindle) Brake off (default: 0)

- 0: Spindle brake on
- 1: Spindle brake off

Cycle run

1 Moves to the starting point at rapid traverse, depending on RB:
- RB not programmed: Moves up to the safety clearance
- RB programmed: Moves to the position RB and then to the safety clearance

2 Drills at reduced feed rate (50%).

3 Moves at feed rate to end of hole

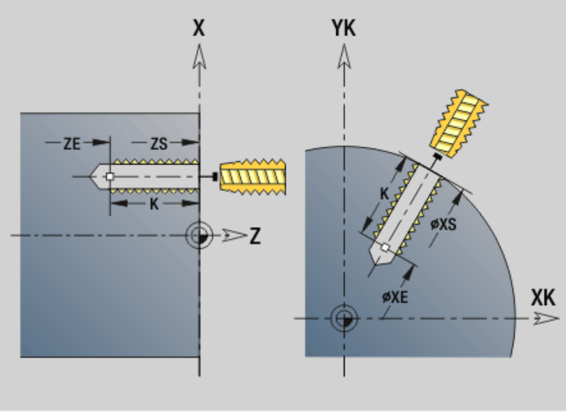
4 Retraction at rapid traverse or feed rate, depending on D.

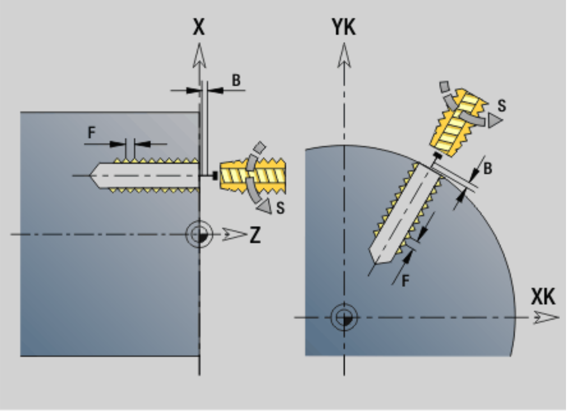
5 Return position depends on RB:
- RB not programmed: Retraction to the starting point
- RB programmed: Retraction to the position RB

Hole pattern: NS refers to the hole contour, and not the definition of the pattern.

Tapping G73

G73 cuts axial/radial threads using driven or stationary tools.





Parameters

ID Drilling contour—Name of the hole definition

NS Block number of contour

- Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)
- No input: Single hole without contour description

XS Starting point of radial hole (diameter value)—single hole without contour description

ZS Starting point of axial hole

Single hole without contour description

XE End point radial hole (diameter value)

Single hole without contour description

ZE End point of axial hole

Single hole without contour description

K Boring depth (hole depth) (alternative to XE/ZE)

Single hole without contour description

F Thread pitch (prevails over the contour description)

B Run-in length

S Retraction speed (default: Shaft speed for tapping)

J Retraction length when using floating tap holders (default: 0)

RB Return plane (radial holes: diameter) – (default: retraction to starting position or to safety clearance)

P Chip breaking depth

I Retraction distance

BS Start element no. (number of the first hole to be machined in a pattern)

BE End element no. (number of the last hole to be machined in a pattern)

H (Spindle) Brake off (default: 0)

- 0: Spindle brake on
- 1: Spindle brake off

The starting position is calculated from the safety clearance and the run-in (slope) length B.

Parameter combinations for single holes without contour description	
XS, XE	ZS, ZE
XS, K	ZS, K
XE, K	ZE, K

Retraction length J: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retraction length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from the taps.

- Hole pattern: NS refers to the hole contour, and not the definition of the pattern.
- Single hole without contour description: Program XS or ZS as alternative.
- Hole with contour description: Do not program XS, ZS.
- Cycle stop interrupts the tapping operation.
- Cycle start resumes the tapping operation.
- Use the feed rate override function for speed changes.
- Spindle override is not effective.
- Use a floating tap holder if the driven tool is not controlled, e.g. by a ROD encoder.

Cycle run

1. Moves at rapid traverse to the starting point:
   - RB not programmed: Moves directly to the starting point
   - RB programmed: Moves to the position RB and then to the starting point
2. Moves along run-in length B feed rate (synchronization of spindle and feed drives).
3. Cuts the thread.
4. Retracts with return speed S:
   - RB not programmed: To the starting point
   - RB programmed: To the position RB

Tapping G36—Single path

G36 cuts axial/radial threads using driven or stationary tools. Depending on X/Z, G36 decides whether a radial or axial thread will be machined.

Move to the starting point before G36. G36 returns to the starting position after having cut the thread.

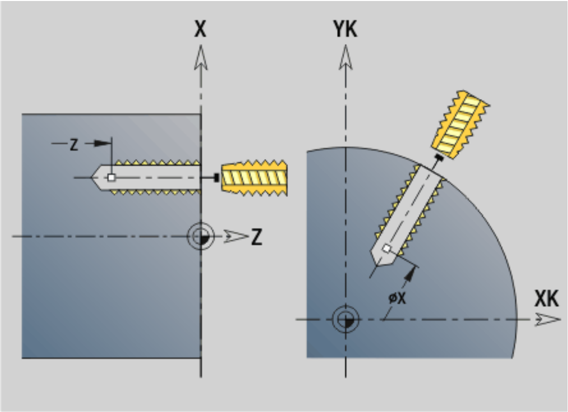
Parameters

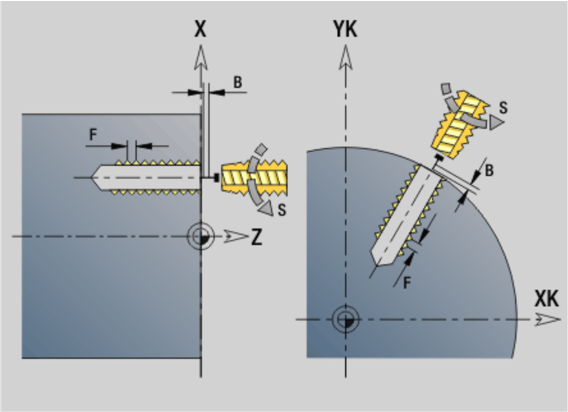
X	End point radial hole (diameter value)
Z	End point of axial hole
F	Feed per revolution (thread pitch)
B	Run-in length for synchronizing spindle and feed drive
S	Retraction speed (default: Shaft speed for tapping)
P	Chip breaking depth
I	Retraction distance

Type of taps:

- Stationary tap: Main spindle and feed drive are synchronized.
- Driven tap: Driven tool and feed drive are synchronized.

- Cycle stop interrupts the tapping operation.
- Cycle start resumes the tapping operation.
- Use the feed rate override function for speed changes.
- Spindle override is not effective.
- Use a floating tap holder if the driven tool is not controlled, e.g. by a ROD encoder.



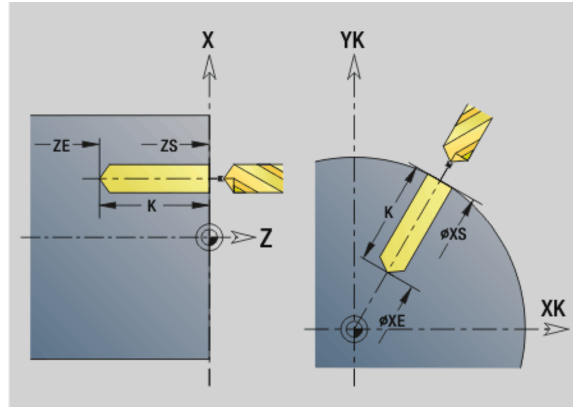


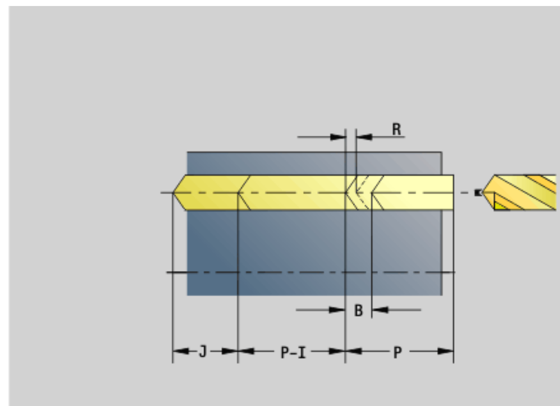
Example: G36

```
. . .
N1 T5 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-30
N4 G14 Q0
N5 T6 G97 S600 M3
N6 G0 X0 Z8
N7 G36 Z-25 F1.5 B3 [tapping]
. . .
```

Deep-hole drilling G74

G74 is used for axial and radial holes in several stages using driven or stationary tools.





Parameters

ID Drilling contour—Name of the hole definition

NS Block number of contour

- Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)
- No input: Single hole without contour description

XS Starting point of radial hole (diameter value)

ZS Starting point of axial hole

XE End point radial hole (diameter value)

ZE End point of axial hole

K Boring depth (hole depth) (alternative to XE/ZE)

P First hole depth

I Reduction value (default: 0)

B Retraction distance (default: to starting point of hole)

J Minimum hole depth (default: 1/10 of P)

R Safety distance (inside)

A Drilling lengths—(default: 0)

V Bore (through-drilling) variant (feed rate reduction 50%)—(default: 0)

- 0: No feed rate reduction
- 1: Feed reduction for through-drilling
- 2: Feed reduction for pre-drilling
- 3: Feed reduction for pre-drilling and through-drilling

RB Retraction plane (radial holes: diameter)—(default: to starting position or to safety clearance)

E Period of dwell for chip breaking at end of hole (in seconds)—(default: 0)

D Retraction speed and infeed within the hole (default: 0)

- 0: Rapid traverse
- 1: Feed rate

BS Start element no. (number of the first hole to be machined in a pattern)

BE End element no. (number of the last hole to be machined in a pattern)

H (Spindle) Brake off (default: 0)

- 0: Spindle brake on
- 1: Spindle brake off

Example: G74

```
. . .
N1 M5
N2 T4 G197 S1000 G195 F0.2 M103
N3 M14
N4 G110 C0
N5 G0 X80 Z2
N6 G745 XK0 YK0 Z2 K80 Wi90 Q4 V2
N7 G74 Z-40 R2 P12 I2 B0 J8 [drilling]
N8 M15
. . .
```

Parameter combinations for single holes without contour description	
XS, XE	ZS, ZE
XS, K	ZS, K
XE, K	ZE, K

The cycle is used for:

- Single hole without contour description
- Hole with contour description (single hole or hole pattern)

"1st drilling depth P" is used for the first pass. MANUALplus then automatically reduces the drilling depth with each subsequent pass by the reduction value I, however, without falling below the minimum drilling depth J. After each pass, the tool is retracted either by retraction distance B or to the starting point of the hole. If the safety distance R is defined, the tool is positioned to this distance at rapid traverse.

Feed rate reduction:

- Indexable insert drill and twist drill with 180° drilling angle
  - A feed rate reduction is only effective if the parameter "Drilling length A" has been defined.
- Other drills
  - Beginning of hole: Feed rate reduction as programmed in V
  - End of hole: Reduction as of hole end point—length of first cut—safety clearance
- Length of first cut = tool tip
- Safety clearance: See user parameter or G47, G147

- Single hole without contour description: Program XS or ZS as alternative.
- Hole with contour description: Do not program XS, ZS.
- Hole pattern: NS refers to the hole contour, and not the definition of the pattern.
- A "feed rate reduction at end" goes into effect only at the last drilling stage.

Cycle run

1 ■ **Hole without contour definition:** Tool is located at the starting point (safety distance from the bore hole).
  ■ **Hole with contour description:** Tool moves at rapid traverse to the starting point:
    ■ RB not programmed: Moves up to the safety clearance
    ■ RB programmed: Moves to the position RB and then to the safety clearance

2 Spot drilling. Feed rate reduction depending on V
3 Drills the hole in several passes
4 Through drilling. Feed rate reduction depending on V
5 Retract at rapid traverse or feed rate, depending on D.
6 Return position depends on RB:
  ■ RB not programmed: Retraction to the starting point
  ■ RB programmed: Retraction to the position RB

Linear pattern, face G743

Cycle G743 is used to machine linear drilling or milling patterns in which the individual features are arranged at a regular spacing on the face.

If the **Final point ZE** has not been defined, the drilling/milling cycle of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with

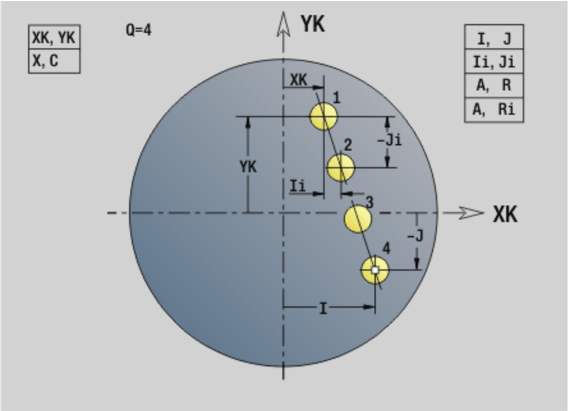
- Drilling cycles (G71, G74, G36)
- The milling cycle for a linear slot (G791)
- The contour milling cycle with "free contour" (G793)

Parameters

XK	Starting point of pattern in Cartesian coordinates
YK	Starting point of pattern in Cartesian coordinates
ZS	Starting point of drilling/milling operation
ZE	Final point of drilling/milling operation
X	Diameter (starting point of pattern in polar coordinates)
C	Angle (starting point of pattern in polar coordinates)
A	Pattern angle
I	Final point of pattern (Cartesian)
Ii	(Final point) Pattern distance (Cartesian)
J	Final point of pattern (Cartesian)
Ji	(Final point) Pattern distance (Cartesian)
R	Length (distance between first and last position)
Ri	Length (distance to next position)
Q	Number of holes/figures—(default: 1)

Parameter combinations for defining the starting point and the pattern positions:

- Starting point of pattern:
  - XK, YK
  - X, C
- Pattern positions:
  - I, J and Q
  - Ii, Ji and Q
  - R, A and Q
  - Ri, Ai and Q



Example: G743

```
%743.nc
[G743]
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G743 XK20 YK5 A45 Ri30 Q2
N6 G791 X50 C0 ZS0 ZE-5 P2 F0.15
N7 M15
END
```

Example: Sequence of commands

```
[Simple drilling pattern]
N.. G743 XK.. YK.. ZS.. ZE.. I.. J.. Q..
. . .
[Drilling pattern with deep-hole drilling]
N.. G743 XK.. YK.. ZS.. I.. J.. Q..
N.. G74 ZE.. P.. I..
. . .
[Milling pattern with linear slot]
N.. G743 XK.. YK.. ZS.. I.. J.. Q..
N.. G791 K.. A.. Z..
. . .
```

Circular pattern, face G745

Cycle G745 is used to machine drilling or milling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the face.

If the **Final point ZE** has not been defined, the drilling/milling cycle of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with

- Drilling cycles (G71, G74, G36)
- The milling cycle for a linear slot (G791)
- The contour milling cycle with "free contour" (G793)

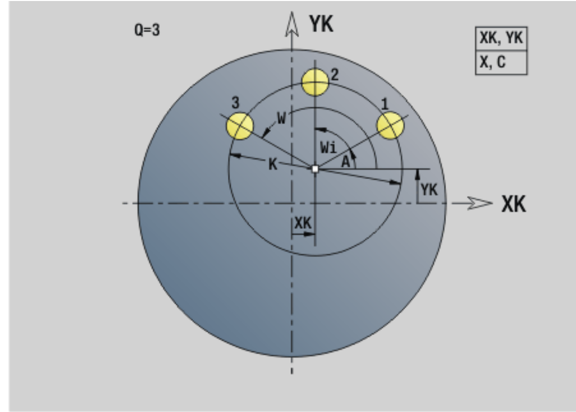
Parameters

XK	Center of pattern in Cartesian coordinates
YK	Center of pattern in Cartesian coordinates
ZS	Starting point of drilling/milling operation
ZE	Final point of drilling/milling operation
X	Diameter (center point of pattern in polar coordinates)
C	Angle (center point of pattern in polar coordinates)
A	Starting angle (position of first hole/figure)
W	Final angle (position of last hole/figure)
Wi	Final angle (distance to the next position)
Q	Number of holes/figures—(default: 1)
V	Rotation direction (default: 0)

- V=0, without W: Figures are arranged on a full circle
- V=0, with W: Figures are arranged on the longer circular arc
- V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- V=1, with W: Clockwise
- V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- V=2, with W: Counterclockwise
- V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)

Parameter combinations for defining the center of the pattern and the pattern positions:

- Center of pattern:
  - X, C
  - XK, YK
- Pattern positions:
  - A, W and Q
  - A, Wi and Q



Example: G745

```
%745.nc
[G745]
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G745 XK0 YK0 K50 A0 Q3
N6 G791 K30 A0 ZS0 ZE-5 P2 F0.15
N7 M15
END
```

Example: Sequence of commands

```
[Simple drilling pattern]
N.. G745 XK.. YK.. ZS.. ZE.. A.. W.. Q..
. . .
[Drilling pattern with deep-hole drilling]
N.. G745 XK.. YK.. ZS.. A.. W.. Q..
N.. G74 ZE.. P.. I..
. . .
[Milling pattern with linear slot]
N.. G745 XK.. YK.. ZS.. ZE.. A.. W.. Q..
N.. G791 K.. A.. Z..
. . .
```

Linear pattern, lateral surface G744

Cycle G744 is used to machine linear drilling patterns or milling patterns in which the individual features are arranged at a regular spacing on the lateral surface.

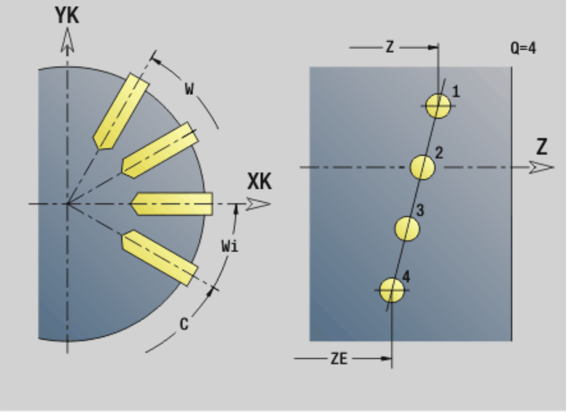
Parameter combinations for defining the starting point and the pattern positions:

- Starting point of pattern: Z, C
- Pattern positions:
  - W and Q
  - Wi and Q

If the **Final point XE** has not been defined, the drilling/milling cycle or the figure definition of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with drilling cycles (G71, G74, G36) or milling cycles (figure definitions with G314, G315, G317).

Parameters

XS	Starting point of drilling/milling operation (diameter value)
Z	Starting point of pattern in polar coordinates
XE	Final point of drilling/milling operation (diameter value)
ZE	Final point of pattern (default: Z)
C	Starting angle of pattern in polar coordinates
W	Final angle of pattern—No input: Holes/figures are arranged on the lateral surface at regular spacing
Wi	Final angle (angle increment), distance to the next position
Q	Number of holes/figures—(default: 1)
A	Angle (orientation angle of the pattern)
R	Length (distance between first and last position [mm], reference: unrolled lateral surface XS)
Ri	Length (distance from the next position [mm], reference: unrolled lateral surface XS)



Example: G744

```
%744.nc
[G744]
N1 T6 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G744 XS102 Z-10 ZE-35 C0 W270 Q5
N6 G71 XS102 K7
N7 M15
END
```

Example: Sequence of commands

```
[Simple drilling pattern]
N.. G744 Z.. C.. XS.. XE.. ZE.. W.. Q..
. . .
[Drilling pattern with deep-hole drilling]
N.. G744 Z.. C.. XS.. XE.. ZE.. W.. Q..
N.. G74 XE.. P.. I..
. . .
[Milling pattern with linear slot]
N.. G744 Z.. C.. XS.. XE.. ZE.. W.. Q..
N.. G792 K.. A.. XS..
. . .
```

Circular pattern, lateral surface G746

Cycle G746 is used to machine drilling patterns or milling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the lateral surface.

Parameter combinations for defining the center of the pattern and the pattern positions:

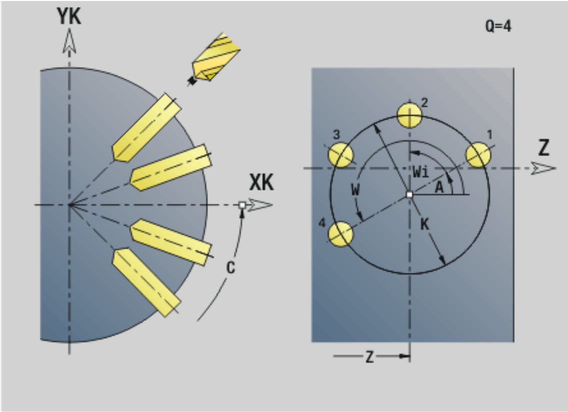
- Center of pattern: Z, C
- Pattern positions:
  - W and Q
  - Wi and Q

If the **Final point XE** has not been defined, the drilling/milling cycle or the figure definition of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with drilling cycles (G71, G74, G36) or milling cycles (figure definitions with G314, G315, G317).

Parameters

Z	Center of pattern in polar coordinates
C	Angle (center point of pattern in polar coordinates)
XS	Starting point of drilling/milling operation (diameter value)
XE	Final point of drilling/milling operation (diameter value)
K	(Pattern) diameter
A	Starting angle (position of first hole/figure)
W	Final angle (position of last hole/figure)
Wi	Final angle (angle increment), distance to the next position
Q	Number of holes/figures—(default: 1)
V	Rotation direction (default: 0)

- V=0, without W: Figures are arranged on a full circle
- V=0, with W: Figures are arranged on the longer circular arc
- V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- V=1, with W: Clockwise
- V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- V=2, with W: Counterclockwise
- V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)



Example: G746

```
%746.nc
[G746]
N1 T6 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G746 Z-40 C0 K40 Q8
N6 G71 XS102 K7
N7 M15
END
```

Example: Sequence of commands

```
[Simple drilling pattern]
N.. G746 Z.. C.. XS.. XE.. K.. A.. W.. Q..
. . .
[Drilling pattern with deep-hole drilling]
N.. G746 Z.. C.. XS..  K.. A.. W.. Q..
N.. G74 XE.. P.. I..
. . .
[Milling pattern with linear slot]
N.. G746 Z.. C.. XS..  K.. A.. W.. Q..
N.. G792 K.. A.. XS..
. . .
```

Thread milling, axial G799

G799 mills a thread in existing holes.

Place the tool on the center of the hole before calling G799. The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one rotation or, with single-point tools, in several rotations.

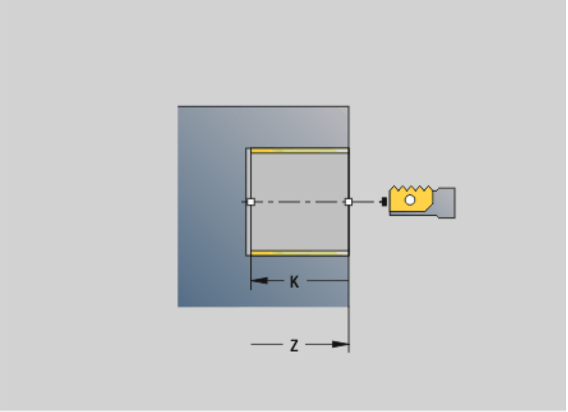
Parameters

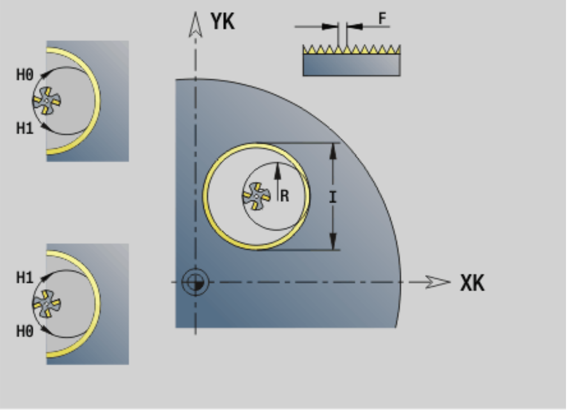
I	Thread diameter
Z	Starting point Z
K	Thread depth
R	Approach radius
F	Thread pitch
J	Direction of thread—(default: 0) ■ 0: Right-hand thread ■ 1: Left-hand thread
H	Cutting direction (default: 0) ■ 0: Up-cut milling ■ 1: Climb milling
V	Milling method ■ 0: The thread is milled in a 360-degree helix ■ 1: The thread is milled in several helical paths (single-point tool)

Use thread-milling tools for cycle G799.

Danger of collision!

Be sure to consider the hole diameter and the diameter of the milling cutter when programming "approach radius R."





Example: G799

```
%799.nc
[G799]
N1 T9 G195 F0.2 G197 S800
N2 G0 X100 Z2
N3 M14
N4 G110 Z2 C45 X100
N5 G799 I12 Z0 K-20 F2 J0 H0
N6 M15
END
```

4.23 C-Axis commands

Reference diameter G120

G120 determines the reference diameter of the unrolled lateral surface. Program G120 if you use CY for G110 to G113. G120 is a modal function.

Parameter
X   Diameter

Example: G120

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100 [reference diameter]
N4 G110 C0
N5 G0 X110 Z5
N6 G41 Q2 H0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635
N12 G40
N13 G110 X105
N14 M15
. . .
```

Zero point shift, C axis G152

G152 defines an absolute zero point for the C axis (reference: Reference point, C axis). The zero point is valid until the end of the program.

Parameter
C   Angle (spindle position) of the new C-axis zero point

Example: G152

```
. . .
N1 M5
N2 T7 G197 S1010 G193 F0.08 M104
N3 M14
N4 G152 C30 [zero point of C axis]
N5 G110 C0
N6 G0 X122 Z-50
N7 G71 X100
N8 M15
. . .
```

Standardize C axis G153

G153 resets a traverse angle >360° or <0° to the corresponding angle modulo 360°—without moving the C axis.

> G153 is only used for lateral-surface machining. An automatic modulo 360° function is carried out on the face.

4.24 Front/rear-face machining

Rapid traverse on front/rear face G100

G100 moves at rapid traverse along the shortest path to the end point.

Parameters

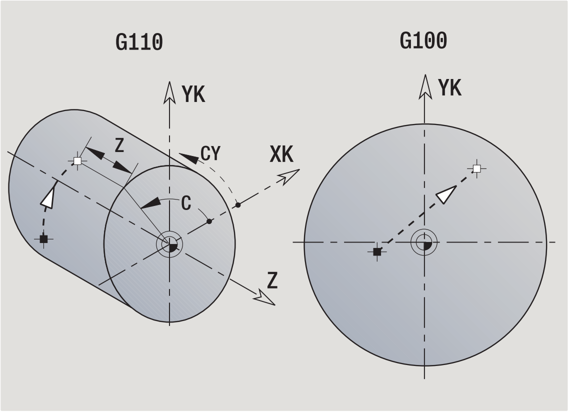
X	End point (diameter)
C	End angle—for angle direction, see graphic support window
XK	End point (Cartesian)
YK	End point (Cartesian)
Z	End point (default: current Z position)

Programming:

- **X, C, XK, YK, Z**: Absolute, incremental or modal
- Program either X–C or XK–YK

Danger of collision!

During G100 the tool moves on a linear path. To position the workpiece to a defined angle, use G110.



Example: G100

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N6 G100 XK20 YK5 [rapid traverse on face]
N7 G101 XK50
N8 G103 XK5 YK50 R50
N9 G101 XK5 YK20
N10 G102 XK20 YK5 R20
N11 G14
N12 M15
. . .
```

Linear segment on front/rear face G101

G101 moves the tool on a linear path at the feed rate to the "end point."

Parameters

X	End point (diameter)
C	End angle—for angle direction, see graphic support window
XK	End point (Cartesian)
YK	End point (Cartesian)
Z	End point (default: current Z position)

Parameters for contour description (G80)

AN Angle to positive XK axis

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

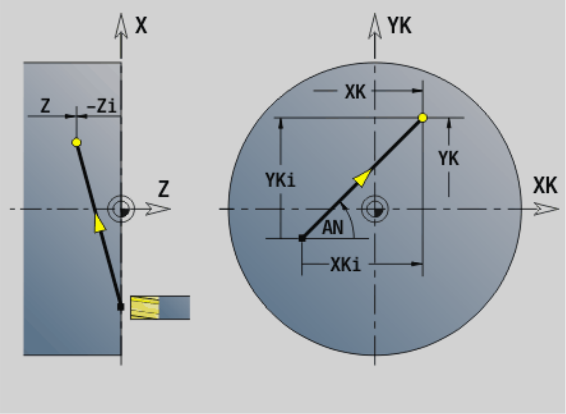
Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

Programming:

- **X, C, XK, YK, Z**: Absolute, incremental or modal
- Program either X–C or XK–YK

Using the parameters AN, BR and Q is only allowed if the contour description is concluded by G80 and used for a cycle.



Example: G101

```
. . .
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G100 XK50 YK0
N6 G1 Z-5
N7 G42 Q1
N8 G101 XK40 [linear path on face]
N9 G101 YK30
N10 G103 XK30 YK40 R10
N11 G101 XK-30
N12 G103 XK-40 YK30 R10
N13 G101 YK-30
N14 G103 XK-30 YK-40 R10
N15 G101 XK30
N16 G103 XK40 YK-30 R10
N17 G101 YK0
N18 G100 XK110 G40
N19 G0 X120 Z50
N20 M15
. . .
```

Circular arc on front/rear face G102/G103

G102/G103 moves the tool in a circular arc at the feed rate to the "end point." The direction of rotation is shown in the graphic support window.

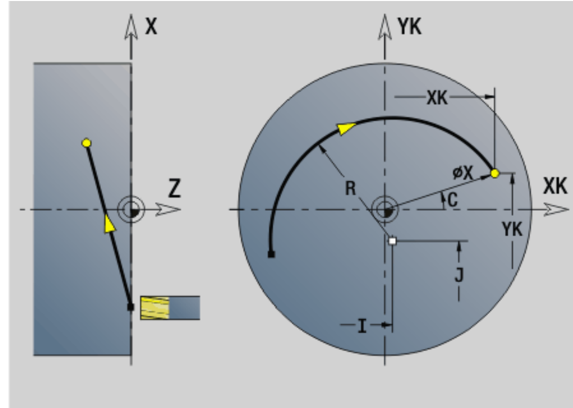
Parameters

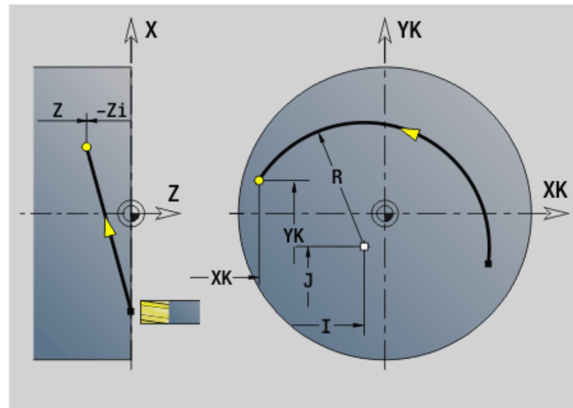
- X End point (diameter)
- C End angle—for angle direction, see graphic support window
- XK End point (Cartesian)
- YK End point (Cartesian)
- R Radius
- I Center point (Cartesian)
- J Center point (Cartesian)
- K Center point for H=2, 3 (Z direction)
- Z End point (default: current Z position)
- H Circular plane (working plane)—(default: 0)
  - H=0, 1: Machining in XY plane (front face)
  - H=2: Machining in YZ plane
  - H=3: Machining in XZ plane

Parameters for contour description (G80)

- AN Angle to positive XK axis
- BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
  - No entry: Tangential transition
  - BR=0: No tangential transition
  - BR>0: Rounding radius
  - BR<0: Width of chamfer
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
  - Q=0: Near point of intersection
  - Q=1: Far point of intersection

> Using the parameters AN, BR and Q is only allowed if the contour description is concluded by G80 and used for a cycle.





Example: G102, G103

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N6 G100 XK20 YK5
N7 G101 XK50
N8 G103 XK5 YK50 R50  [circular arc]
N9 G101 XK5 YK20
N10 G102 XK20 YK5 R20
N12 M15
. . .
```

If you program H=2 or H=3, you can machine linear slots with a circular base. If

- H=2: Define the circle center with I and K.
- H=3: Define the circle center with J and K.

Programming:

- **X, C, XK, YK, Z**: Absolute, incremental or modal
- **I, J, K:** Absolute or incremental
- Program either X–C or XK–YK
- Program either center or radius
- For radius: Only arcs <= 180° are possible
- End point in the coordinate origin: Program XK=0 and YK=0.

4.25 Lateral surface machining

Rapid traverse, lateral surface G110

G110 moves at rapid traverse along the shortest path to the end point.

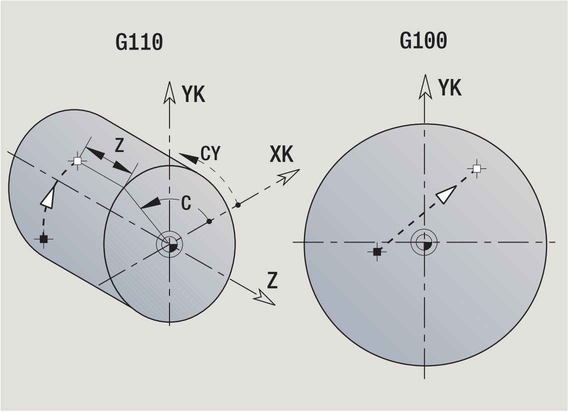
G110 is recommended for **positioning the C axis** to a defined angle (programming: N.. G110 C...).

Parameters

Z	End point
C	End angle
CY	End point as linear value (referenced to unrolled reference diameter G120)
X	End point (diameter)

Programming:

- **Z, C, CY:** Absolute, incremental, or modal
- Program either Z–C or Z–CY



Example: G110

```
. . .
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0 [rapid, lateral surface]
N5 G0 X110 Z5
N6 G110 Z-20 CY0
N7 G111 Z-40
N8 G113 CY39.2699 K-40 J19.635
N9 G111 Z-20
N10 G113 CY0 K-20 J19.635
N11 M15
. . .
```

Line segment on lateral surface G111

G111 moves the tool on a linear path at the feed rate to the "end point."

Parameters

- Z — End point
- C — End angle—for angle direction, see graphic support window
- CY — End point as linear value (referenced to unrolled reference diameter G120)
- X — End point (diameter value) – (default: current X position)

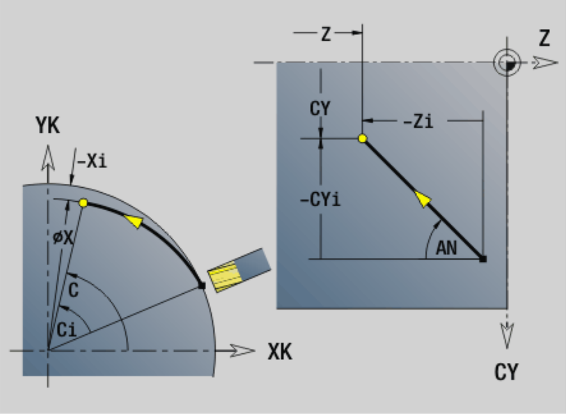
Parameters for contour description (G80)

- AN — Angle to positive Z axis
- BR — Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
  - No entry: Tangential transition
  - BR=0: No tangential transition
  - BR>0: Rounding radius
  - BR<0: Width of chamfer
- Q — Point of intersection. End point if the line segment intersects a circular arc (default: 0):
  - Q=0: Near point of intersection
  - Q=1: Far point of intersection

Using the parameters AN, BR and Q is only allowed if the contour description is concluded by G80 and used for a cycle.

Programming:

- **Z, C, CY:** Absolute, incremental, or modal
- Program either Z–C or Z–CY



Example: G111

```
. . .
[G111, G120]
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G41 Q2 H0
N7 G110 Z-20 CY0
N8 G111 Z-40 [linear path on lateral surface]
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635
N12 G40
N13 G110 X105
N14 M15
. . .
```

Circular arc on lateral surface G112/G113

G112/G113 moves the tool in a circular arc at the feed rate to the "end point."

Parameters

- Z End point
- C End angle—for angle direction, see graphic support window
- CY End point as linear value (referenced to unrolled reference diameter G120)
- R Radius
- K Center
- J Center point as linear value (referenced to unrolled G120 reference diameter)
- W Center of angle (angular direction: see help graphic)
- X End point (diameter value) – (default: current X position)

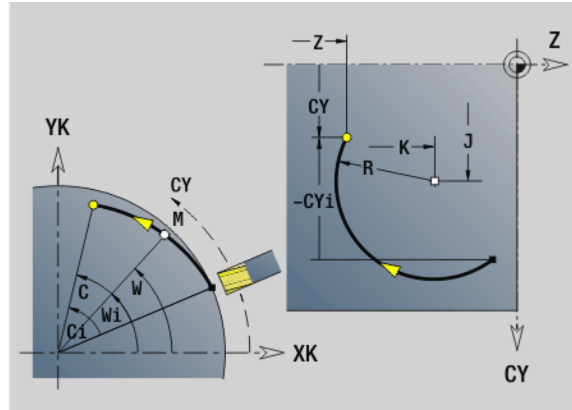
Parameters for contour description (G80)

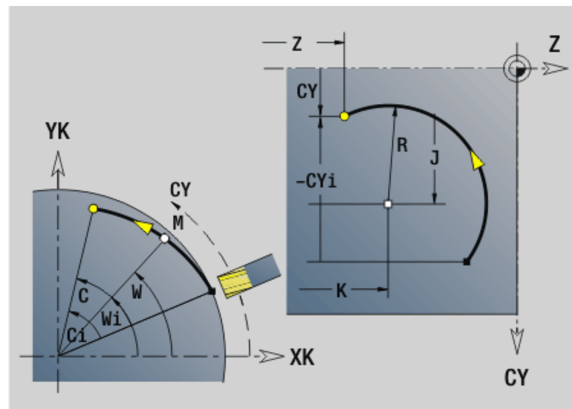
- AN Angle to positive Z axis
- BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
  - No entry: Tangential transition
  - BR=0: No tangential transition
  - BR>0: Rounding radius
  - BR<0: Width of chamfer
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
  - Q=0: Near point of intersection
  - Q=1: Far point of intersection

Using the parameters AN, BR and Q is only allowed if the contour description is concluded by G80 and used for a cycle.

Programming:

- **Z, C, CY:** Absolute, incremental, or modal
- **K; W, J:** Absolute or incremental
- Program either Z–C or Z-CY **and** K–J.
- Program either center or radius
- For radius: Only arcs <= 180° are possible





Example: G112, G113

```
. . .
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635 [circular arc]
N10 G111 Z-20
N11 G112 CY0 K-20 J19.635
N13 M15
```

4.26 Milling cycles

Overview of milling cycles

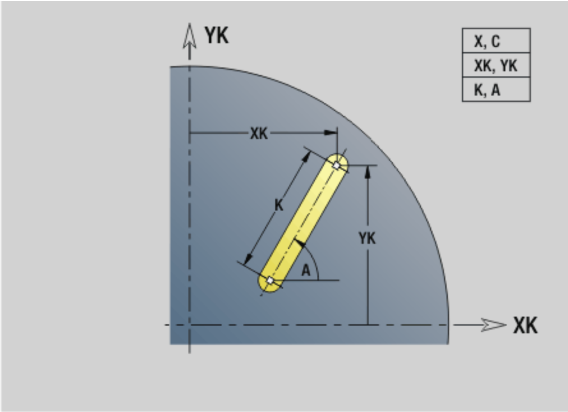
- G791 Linear slot on the face. The position and length of the slot are defined directly in the cycle; slot width = cutter diameter: Page 339
- G792 Linear slot on the lateral surface. The position and length of the slot are defined directly in the cycle; slot width = cutter diameter: Page 340
- G793 Contour and figure milling cycle on the face. The contour is described directly after the cycle and concluded by G80 (compatibility cycle MANUALplus 4110): Page 341
- G794 Contour and figure milling cycle on the lateral surface. The contour is described directly after the cycle and concluded by G80 (compatibility cycle MANUALplus 4110): Page 343
- G797 Face milling. Mills figures (circles, polygons, individual surfaces, contours) as islands on the face: Page 345
- G798 Helical slot milling. Mills a helical slot on the lateral surface, slot width = cutter diameter: Page 347
- G840 Contour milling. Mills ICP contours and figures. Closed contours are machined inside/outside of the contour, or on the contour. Open contours are machined from the left/right of the contour, or on the contour. G840 is used on the face and lateral surface: Page 348
- G845 Pocket milling—roughing. Roughs out closed ICP contours and figures on the face and lateral surface: Page 358
- G846 Pocket milling—finishing. Finishes closed ICP contours and figures on the face and lateral surface: Page 364

Contour definitions in the MACHINING section (figures)

- Face
  - G301 Linear slot: Page 227
  - G302/G303 Circular slot: Page 227
  - G304 Full circle: Page 228
  - G305 Rectangle: Page 228
  - G307 Eccentric polygon: Page 229
- Lateral surface
  - G311 Linear slot: Page 236
  - G312/G313 Circular slot: Page 236
  - G314 Full circle: Page 237
  - G315 Rectangle: Page 237
  - G317 Eccentric polygon: Page 238

Linear slot on face G791

G791 mills a slot from the current tool position to the end point. The slot width equals the diameter of the milling cutter. Oversizes are not taken into account.

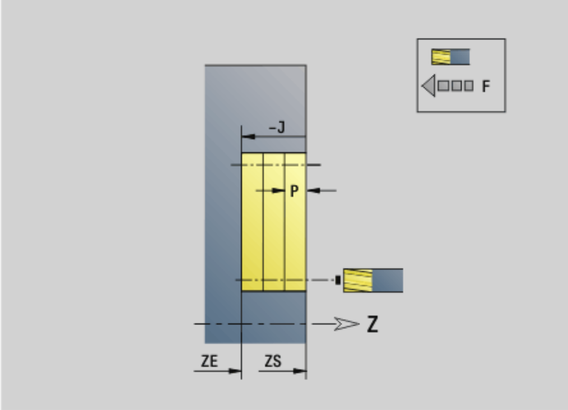


Parameters

X Final point of slot in polar coordinates (diameter)
C Final angle. Final point of slot in polar coordinates (for angle direction, see help graphic)
XK Final point of slot (Cartesian)
YK Final point of slot (Cartesian)
K Slot length referenced to center of cutter
A Slot angle (reference: see help graphic)
ZE Milling floor
ZS Milling top edge
J Milling depth

- J>0: Infeed direction –Z
- J<0: Infeed direction +Z

P Maximum approach (default: total depth in one infeed)
F Approach feed (infeed rate) (default: active feed rate)



Parameter combinations for definition of the end point: see help graphic

Parameter combinations for definition of the milling plane:

- Milling floor ZE, milling top edge ZS
- Milling floor ZE, milling depth J
- Milling top edge ZS, milling depth J
- Milling floor ZE

- Rotate the spindle to the desired angle position **before** calling G791.
- If you use a spindle positioning device (no C axis), an axial slot is machined centrically to the rotary axis.
- If J or ZS is defined, the tool approaches to safety clearance in Z and then mills the slot. If J and ZS are not defined, the milling cycle starts from the current tool position.

Example: G791

```
%791.nc
[G791]
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G100 XK20 YK5
N6 G791 XK30 YK5 ZE-5 J5 P2
N7 M15
END
```

Linear slot on lateral surface G792

G792 mills a slot from the current tool position to the end point. The slot width equals the diameter of the milling cutter. Oversizes are not taken into account.

Parameters

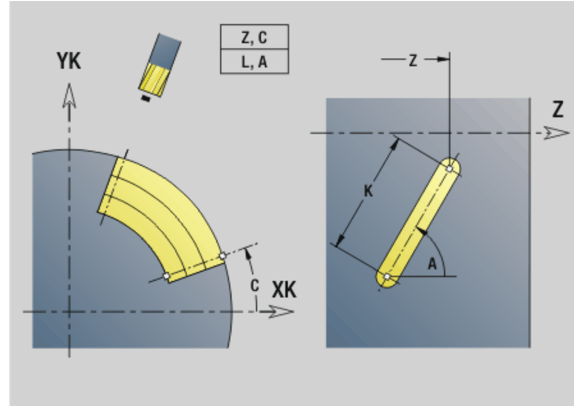
Z	Final point of slot
C	Final angle. Final point of slot (for angle direction, see help graphic)
K	Slot length referenced to center of cutter
A	Slot angle (reference: see help graphic)
XE	Milling floor
XS	Milling top edge
J	Milling depth ■ J>0: Infeed direction –X ■ J<0: Infeed direction +X
P	Maximum approach (default: total depth in one infeed)
F	Approach feed (infeed rate) (default: active feed rate)

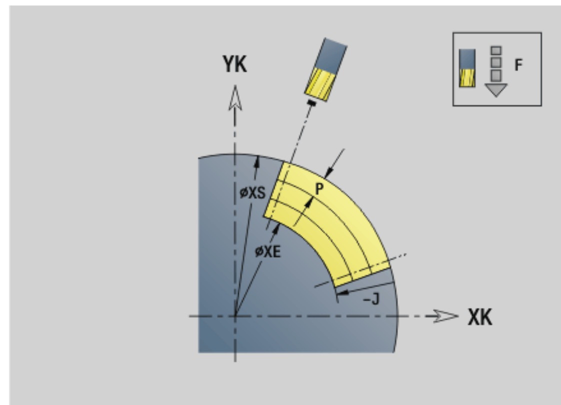
Parameter combinations for definition of the end point: see help graphic

Parameter combinations for definition of the milling plane:

- Milling floor XE, milling top edge XS
- Milling floor XE, milling depth J
- Milling top edge XS, milling depth J
- Milling floor XE

- Rotate the spindle to the desired angle position **before** calling G792.
- If you use a spindle positioning device (no C axis), a radial slot is machined parallel to the Z axis.
- If J or XS is defined, the tool approaches to safety clearance in X and then mills the slot. If J and XS are not defined, the milling cycle starts from the current tool position.





Example: G792

```
%792.nc
[G792]
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z5
N5 G0 X102 Z-30
N6 G792 K25 A45 XE97 J3 P2 F0.15
N7 M15
END
```

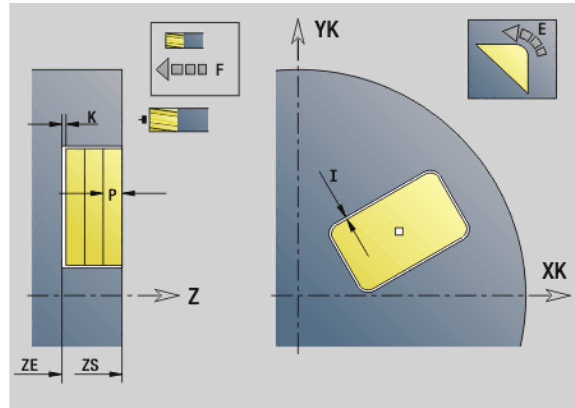
Contour and figure milling cycle, face G793

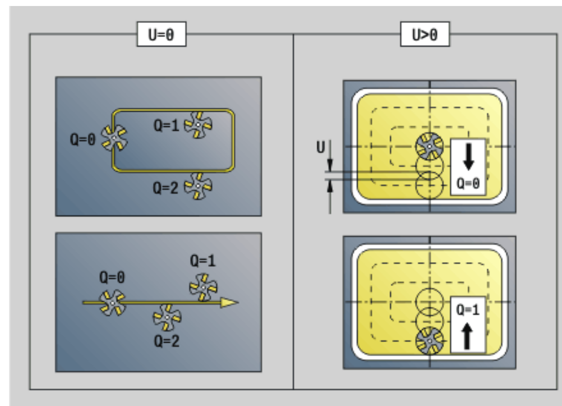
G793 mills figures or (open or closed) "free" contours.

G793 is followed by:

- **The figure to be milled** with:
  - Contour definition of the figure (G301 to G307)—See "Front and rear face contours" on page 223.
  - Conclusion of milling contour (G80)
- **The free contour** with:
  - Starting point of milling contour (G100)
  - Milling contour (G101, G102, G103)
  - Conclusion of milling contour (G80)

> Preferentially use ICP and the G840, G845 and G846 cycles to program the contour description in the geometry section.





Parameters

ZS Milling top edge

ZE Milling floor

P Maximum approach (default: total depth in one infeed)

U Overlap factor—contour milling or pocket milling (default: 0)

- U=0: Contour milling
- U>0: Pocket milling—minimum overlap of milling paths = U*milling diameter

R Approach radius (radius of approaching/departing arc)—(default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially

I Contour-parallel oversize

K Oversize Z

F Infeed rate

E Reduced feed rate for circular elements (default: current feed rate)

H Cutting direction (default: 0): The **cutting direction** can be changed with H and the direction of tool rotation.

- 0: Up-cut milling
- 1: Climb milling

Parameters

Q Cycle type (default: 0): Depending on U, the following applies:

- **Contour milling (U=0)**
  - Q=0: Center of milling cutter on the contour
  - Q=1, closed contour: Inside milling
  - Q=1, open contour: Left in machining direction
  - Q=2, closed contour: Outside milling
  - Q=2, open contour: Right in machining direction
  - Q=3, open contour: Milling location depends on "H" and the direction of tool rotation—see help graphic
- **Pocket milling (U>0)**
  - Q=0: From the inside toward the outside
  - Q=1: From the outside toward the inside

O Roughing/finishing

- 0: Roughing. With each infeed, the complete surface is machined.
- 1: Finishing. The surface is machined with the last infeed. In all previous infeeds, the cycle machines only the contour.

- **Milling depth:** The cycle calculates the depth from the `Milling top edge` and the `Milling floor`—taking the oversizes into account.
- **Milling cutter radius compensation:** Effective (except for contour milling with Q=0).
- **Approach and departure:** For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For contour milling and finishing (pocket milling), define with the `Approach radius` whether the tool is to approach directly or in an arc.
- **G57/G58 oversizes** are taken into account if the `Oversizes I, K` are not programmed:
  - G57: Oversize in X, Z direction
  - G58: The oversize "shifts" the milling contour as follows:
    – With inside milling and closed contour: The contour is contracted
    – With outside milling and closed contour: The contour is expanded
    – With open contour and Q=1: Left in machining direction
    – With open contour and Q=2: Right in machining direction

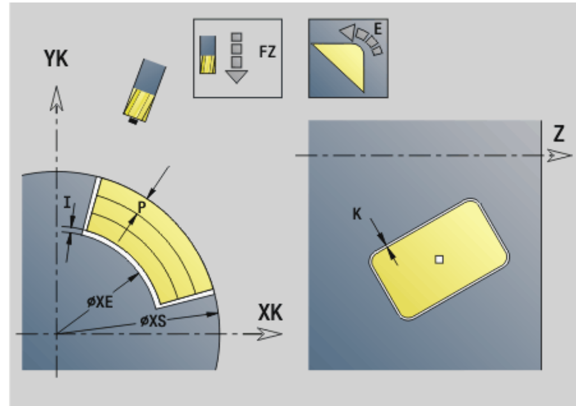
Contour and figure milling cycle, lateral surface G794

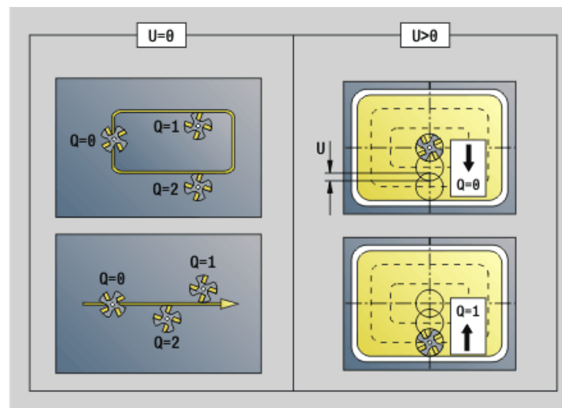
G794 mills figures or (open or closed) "free" contours.

G794 is followed by:

- **The figure to be milled** with:
  - Contour definition of the figure (G311 to G317)—See "Lateral surface contours" on page 232.
  - Conclusion of contour definition (G80)
- **The free contour** with:
  - Starting point (G110)
  - Contour definition (G111, G112, G113)
  - Conclusion of contour definition (G80)

> Preferentially use ICP and the G840, G845 and G846 cycles to program the contour description in the geometry section.





Parameters

XS Milling top edge (diameter value)

XE Milling floor (diameter)

P Maximum approach (default: total depth in one infeed)

U Overlap factor—contour milling or pocket milling (default: 0)

- U=0: Contour milling
- U>0: Pocket milling—minimum overlap of milling paths = U*milling diameter

R Approach radius (radius of approaching/departing arc)—(default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially

I Oversize X

K Contour-parallel oversize

F Infeed rate

E Reduced feed rate for circular elements (default: current feed rate)

H Cutting direction (default: 0): The **cutting direction** can be changed with H and the direction of tool rotation.

- 0: Up-cut milling
- 1: Climb milling

Example: G794

```
%314_G315.nc
[G314 / G315]
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z5
N5 G794 XS100 XE97 P2 U0.5 R0 K0.5 F0.15
N6 G314 Z-35 C0 R20
N7 G80
N8 M15
END
```

Parameters

Q Cycle type (default: 0): Depending on U, the following applies:

- **Contour milling (U=0)**
  - Q=0: Center of milling cutter on the contour
  - Q=1, closed contour: Inside milling
  - Q=1, open contour: Left in machining direction
  - Q=2, closed contour: Outside milling
  - Q=2, open contour: Right in machining direction
  - Q=3, open contour: Milling location depends on "H" and the direction of tool rotation—see help graphic
- **Pocket milling (U>0)**
  - Q=0: From the inside toward the outside
  - Q=1: From the outside toward the inside

O Roughing/finishing

- 0: Roughing. With each infeed, the complete surface is machined.
- 1: Finishing. The surface is machined with the last infeed. In all previous infeeds, the cycle machines only the contour.

- **Milling depth:** The cycle calculates the milling depth from the **`Milling top edge`** and the **`Milling floor`**—taking the oversizes into account.
- **Milling cutter radius compensation:** Effective (except for contour milling with Q=0).
- **Approach and departure:** For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For contour milling and finishing (pocket milling), define with the **`Approach radius`** whether the tool is to approach directly or in an arc.
- **G57/G58 oversizes** are taken into account if the **`Oversizes I, K`** are not programmed:
  - G57: Oversize in X, Z direction
  - G58: The oversize "shifts" the milling contour as follows:
    – With inside milling and closed contour: The contour is contracted
    – With outside milling and closed contour: The contour is expanded
    – With open contour and Q=1: Left in machining direction
    – With open contour and Q=2: Right in machining direction

Area milling, face G797

Depending on Q, G797 mills surfaces, a polygon, or the figure defined in the command following G797.

Parameters

X	Limit diameter
ZS	Milling top edge
ZE	Milling floor
B	Width across flats (omit for Q=0): B defines the remaining material. For an even number of surfaces, you can program B as an alternative to V.

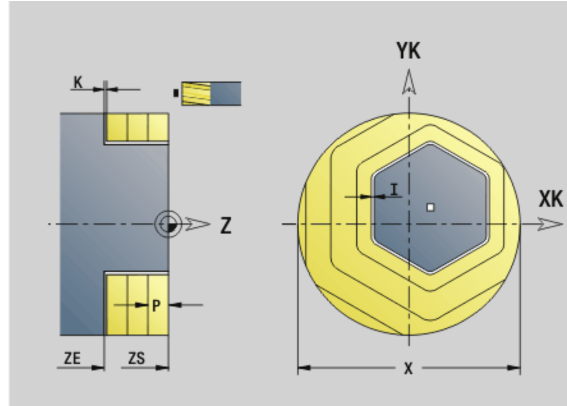
- Q=1: B=Residual depth
- Q>=2: B=Width across flats

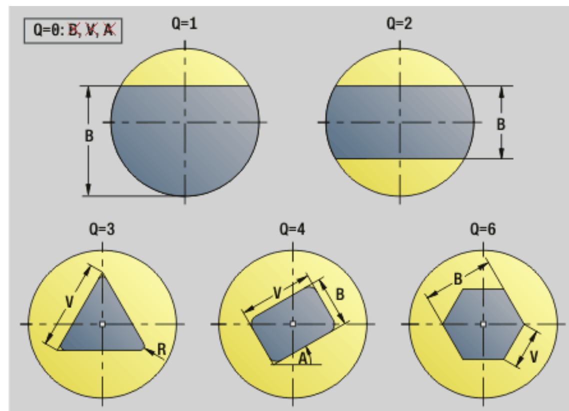
V	Edge length (omitted for Q=0)
R	Chamfer/rounding
A	Inclination angle (reference: see help graphic)—omitted for Q=0
Q	Number of surfaces (default: 0): Range 0 <= Q <= 127

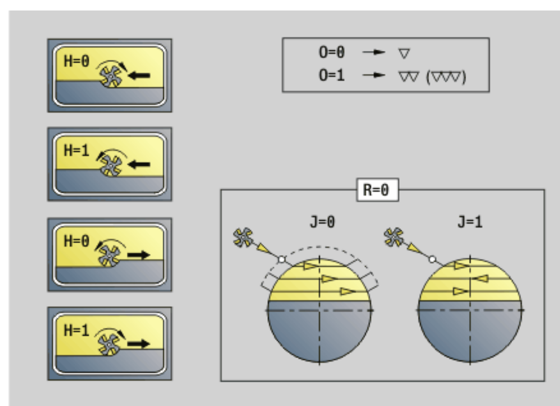
- Q=0: G797 is followed by a figure definition (G301 to G307, G80) or a closed contour definition (G100 to G103, G80)
- Q=1: One surface
- Q=2: Two surfaces offset by 180°
- Q=3: Triangle
- Q=4: Rectangle, square
- Q>4: Polygon

P	Maximum approach (default: total depth in one infeed)
U	Overlap factor (default: 0.5): Minimum overlap of milling paths = U*milling diameter
I	Contour-parallel oversize
K	Oversize Z
F	Infeed rate
E	Reduced feed rate for circular elements (default: current feed rate)
H	Cutting direction (default: 0): The **cutting direction** can be changed with H and the direction of tool rotation (see help graphic)

- 0: Up-cut milling
- 1: Climb milling







Parameters

O Roughing/finishing

- 0: Roughing. With each infeed, the complete surface is machined.
- 1: Finishing. The surface is machined with the last infeed. In all previous infeeds, the cycle machines only the contour.

J Milling direction. For polygons without chamfers/rounding arcs, J defines whether a unidirectional or bidirectional milling operation is to be executed (see help graphic).

- 0: Unidirectional
- 1: Bidirectional

Programming notes:

The cycle calculates the milling depth from ZS and ZE, taking the oversizes into account.

Surfaces and figures defined with G797 (Q>0) are symmetric with respect to the center. A figure defined in the following command can be **outside the center.**

G797 Q0 .. is followed by:

- **The figure to be milled** with:
  - Contour definition of the figure (G301 to G307)—See "Front and rear face contours" on page 223.
  - Conclusion of milling contour (G80)
- **The free contour** with:
  - Starting point of milling contour (G100)
  - Milling contour (G101, G102, G103)
  - Conclusion of milling contour (G80)

Example: G797

```
%797.nc
[G797]
N1 T9 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G797 X100 Z0 ZE-5 B50 R2 A0 Q4 P2 U0.5
N6 G100 Z2
N7 M15
END
```

Example: G797/G304

```
%304_G305.nc
[G304]
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G797 X100 ZS0 ZE-5 Q0 P2 F0.15
N6 G304 XK20 YK5 R20
N7 G80
N4 G0 X100 Z2
N5 G797 X100 ZS0 ZE-5 Q0 P2 F0.15
N6 G305 XK20 YK5 R6 B30 K45 A20
N7 G80
N8 M15
END
```

Helical-slot milling G798

G798 mills a helical slot from the current tool position to the **Final point X, Z**. The slot width equals the diameter of the milling cutter.

Parameters

X End point (diameter value) – (default: current X position)
Z Final point of slot
C Starting angle
F Thread pitch:

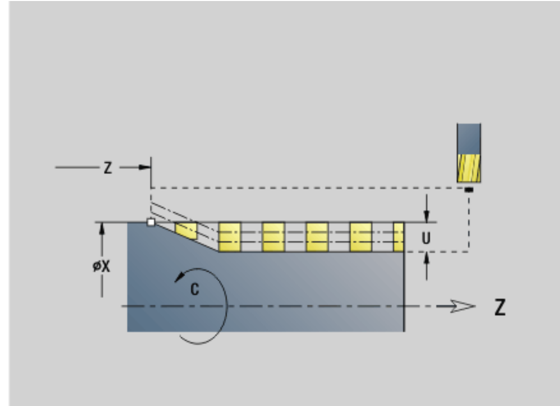
- F positive: Right-hand thread
- F negative: Left-hand thread

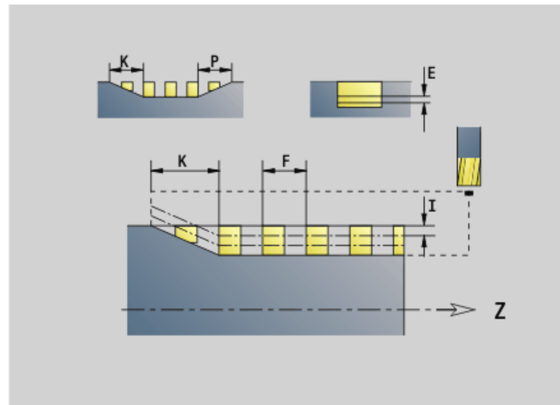
P Slop. length (run-in length)—ramp at the beginning of the slot (default: 0)
K End. length (run-out length)—ramp at the end of the slot (default: 0)
U Thread depth
I Maximum approach (default: total depth in one infeed)
E Reduction value for infeed reduction (default: 1)
D No. of gears (threads per unit)

Infeeds:

- **Infeed I** is used for the first infeed movement.
- The Control calculates all subsequent infeed movements as follows:
  Current infeed = I * (1 – (n–1) * E)
  (n: nth infeed)
- The infeed movement is reduced down to >= 0.5 mm. Following that, each infeed movement will amount to 0.5 mm.

You can mill a helical slot only from the outside.





Example: G798

```
%798.nc
[G798]
N1 T9 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X80 Z15
N5 G798 X80 Z-120 C0 F20 K20 U5 I1
N6 G100 Z2
N7 M15
END
```

Contour milling G840

G840—Fundamentals

G840 mills or deburrs open or closed contours (figures or "free contours").

Plunge strategies: Depending on the cutter you are using, select one of the following strategies:

- **Vertical plunge:** The cycle moves the tool to the starting point; the tool plunges and mills the contour.
- **Calculate positions, predrill, mill.** The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with "G840 A1 ..".
  - Predrill with "G71 NF .."
  - Call cycle "G840 A0 ..". The cycle positions the tool above the hole; the tool plunges and mills the contour.
- **Predrilling, milling.** The machining process is performed in the following steps:
  - Predrill with "G71 .."
  - Position the cutter above the hole. Call cycle "G840 A0 ..". The cycle plunges and mills the contour or contour section.

If the milling contour consists of multiple sections, G840 takes all the sections of the contour into account for predrilling and milling. Call "G840 A0 .." separately for each section when calculating the hole positions without "G840 A1 ..".

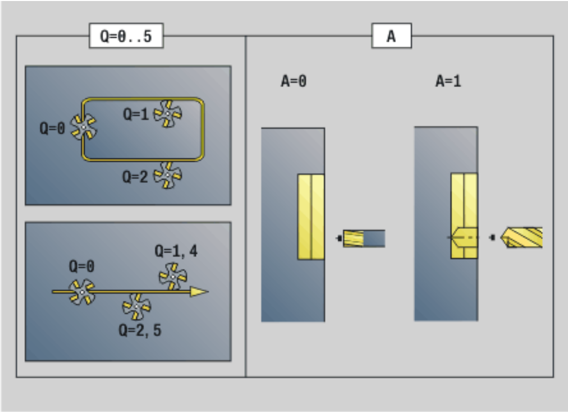
Oversize: A G58 oversize "shifts" the contour to be milled in the direction given in cycle type **Q**.

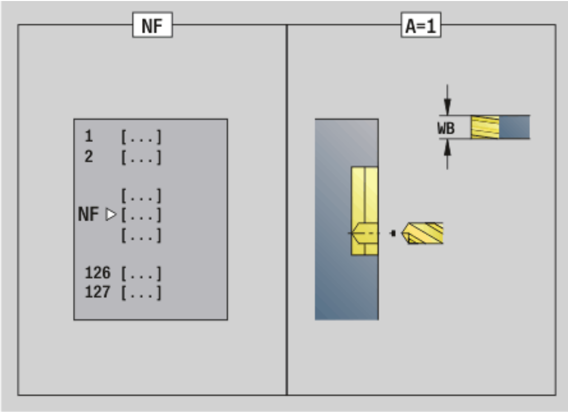
- With inside milling and closed contour: Shifted inward
- With outside milling and closed contour: Shifted outward
- Open contour: Shifts to the left or right depending on Q

- If Q=0, oversizes are not taken into account.
- G57 and negative G58 oversizes are not taken into account.

G840—Calculating hole positions

"G840 A1 .." calculates the hole positions and stores them at the reference specified in "NF." Program only the parameters given in the following table.





See also:

- G840—Fundamentals: Page 348
- G840—Milling: Page 351

Parameters—Calculating hole positions

Q Cycle type (= milling location)

- Open contour. If there is any overlapping, Q defines whether the first section (as of starting point) or the entire contour is to be machined.
  - Q=0: Center of milling cutter on the contour (hole position = starting point)
  - Q=1: Machining at the left of the contour. If there is overlapping, only the first area of the contour is machined.
  - Q=2: Machining at the right of the contour. If there is overlapping, only the first area of the contour is machined.
  - Q=3: Not allowed
  - Q=4: Machining at the left of the contour. If there is overlapping, the entire contour is machined.
  - Q=5: Machining at the right of the contour. If there is overlapping, the entire contour is machined.
- Closed contours
  - Q=0: Center of milling cutter on the contour (hole position = starting point)
  - Q=1: Inside milling
  - Q=2: Outside milling
  - Q=3 to 5: Not allowed

ID Milling contour—name of the contour to be milled

NS Contour start block number—beginning of contour section

- Figures: Block number of the figure
- Free closed contour: First contour element (not starting point)
- Open contour: First contour element (not starting point)

NE Contour end block number—end of contour section

- Figures, free closed contour: No input
- Open contour: last contour element
- Contour consists of one element:
  - No input: Machining in contour direction
  - NS=NE programmed: Machining against the contour direction

Parameters—Calculating hole positions

D Starting element number for partial figures

The direction of contour definition for figures is counterclockwise. The first contour element for figures:

- Circular slot: The larger arc
- Full circle: The upper semicircle
- Rectangles, polygons and linear slots: The orientation angle points to the first contour element.

V Ending element number for partial figures

A Sequence for "Calculate hole positions": A=1

NF Position mark—reference at which the cycle stores the hole positions [1 to 127].

WB Rework diameter—diameter of the milling cutter

Program D and V to machine parts of a figure.

- The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling "G840 A1 ..".
- Program oversizes for calculating the hole positions **and** for milling.

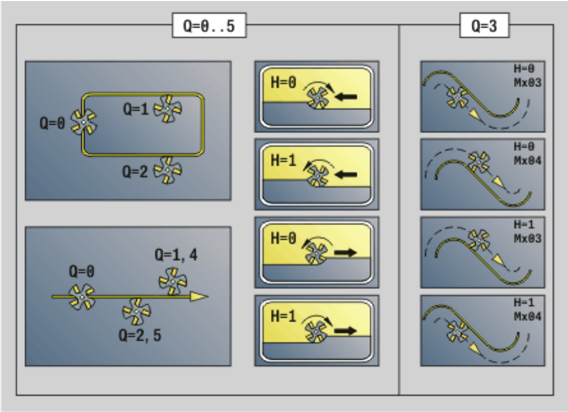
G840 overwrites any hole positions that may still be stored at the reference "NF."

G840—Milling

You can change the machining direction and the cutter radius compensation (TRC) with the **`cycle type Q`**, the **`cutting direction H`** and the rotational direction of the tool (see following table). Program only the parameters given in the following table.

See also:

- G840—Fundamentals: Page 348
- G840—Calculating hole positions: Page 349



Parameters—Milling

Q Cycle type (= milling location).

- Open contour. If there is any overlapping, Q defines whether the first section (as of starting point) or the entire contour is to be machined.
  - Q=0: Center of milling cutter on the contour (without TRC)
  - Q=1: Machining at the left of the contour. If there is any overlapping, G840 machines only the first section of the contour (starting point: 1st point of intersection).
  - Q=2: Machining at the right of the contour. If there is any overlapping, G840 machines only the first section of the contour (starting point: 1st point of intersection).
  - Q=3: The contour is machined to the left or right depending on H and the direction of cutter rotation (see table). If there is any overlapping, G840 machines only the first section of the contour (starting point: 1st point of intersection).
  - Q=4: Machining at the left of the contour. If there is overlapping, G840 machines the entire contour.
  - Q=5: Machining at the right of the contour. If there is overlapping, G840 machines the entire contour.
- Closed contours
  - Q=0: Center of milling cutter on the contour (hole position = starting point)
  - Q=1: Inside milling
  - Q=2: Outside milling
  - Q=3 to 5: Not allowed

ID Milling contour—name of the contour to be milled

NS Block number—beginning of contour section

- Figures: Block number of the figure
- Free open or closed contour: First contour element (not starting point)

Parameters—Milling

NE Block number—end of contour section

- Figures, free closed contour: No input
- Free open contour: Last contour element
- Contour consists of one element:
  - No input: Machining in contour direction
  - NS=NE programmed: Machining against the contour direction

H Cutting direction (default: 0)

- 0: Up-cut milling
- 1: Climb milling

I (Maximum) infeed (default: milling in one infeed)

F Infeed rate (depth infeed) – (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Contour element is approached/departed tangentially on a line

P Milling depth (default: depth from the contour description)

XS Milling top edge—lateral surface (replaces the reference plane from the contour definition)

ZS Milling top edge—face (replaces the reference plane from the contour definition)

RB Retraction plane (default: back to starting position)

- Front or read face: Return position in Z direction
- Lateral surface: Return position in X direction (diameter)

D Starting element number when partial figures are machined.

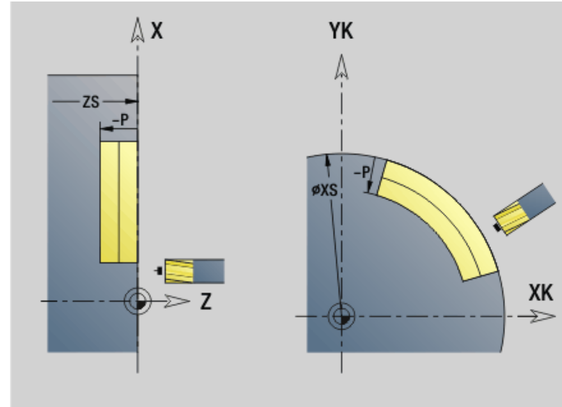
V End element number when partial figures are machined.

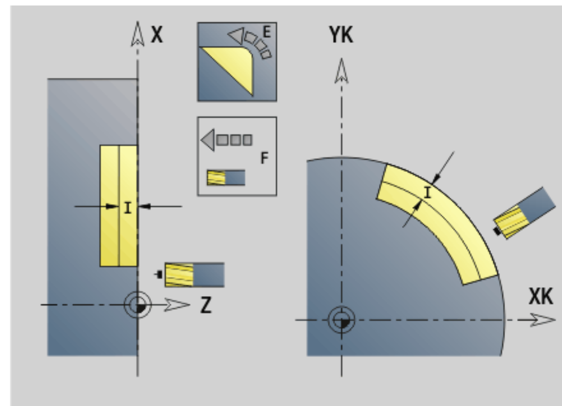
The direction of contour definition for figures is counterclockwise. The first contour element for figures:

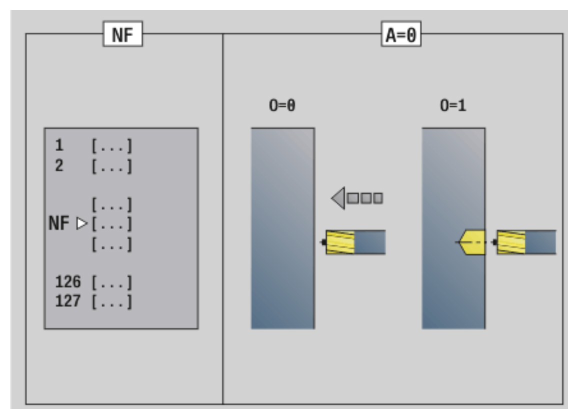
- Circular slot: The larger arc
- Full circle: The upper semicircle
- Rectangles, polygons and linear slots: The orientation angle points to the first contour element.

A Sequence for "Milling, deburring": A=0 (default=0)

NF Position mark—reference from which the cycle reads the hole positions [1 to 127].







Parameters—Milling

O Plunging behavior (default: 0)

- O=0: Vertical plunging
- O=1: With predrilling
  - If NF is programmed: The cycle positions the milling cutter above the first hole position saved in NF, then plunges and mills the first section. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next section, etc.
  - If NF is not programmed: The tool plunges at the current position and mills the section. If required, repeat this operation for the next section, etc.

Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For figures, use D and V to select the approach/departure element.

Cycle run for milling

1 Starting position (X, Z, C) is the position before the cycle begins.

2 Calculates the milling depth infeeds.

3 Approaches to safety clearance.

- If O=0: Infeed to the first milling depth.
- If O=1: Plunges to the first milling depth.

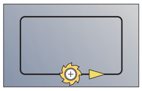
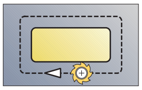
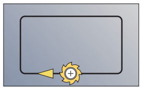
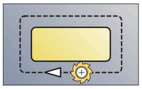
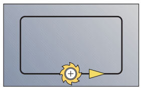
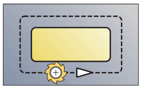
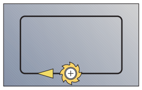
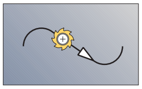
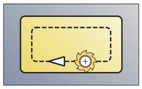
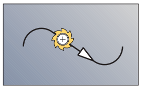
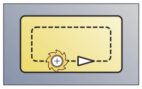
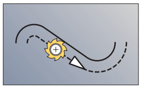
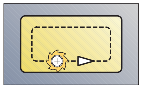
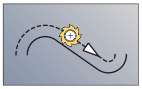
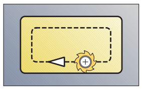
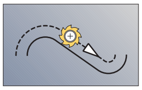
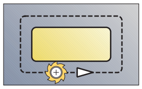
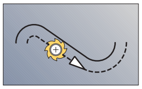
4 Mills the contour.

5
- For open contour and slots with slot width equal to the cutter diameter: Moves at feed rate to the next milling depth, or plunges at rapid to the next milling depth and mills the contour in reverse direction.
- For closed contours and slots: Lifts off by the safety clearance, moves forward and approaches to the next milling depth, or plunges to the next milling depth.

6 Repeats steps 4 and 5 until the complete contour is milled.

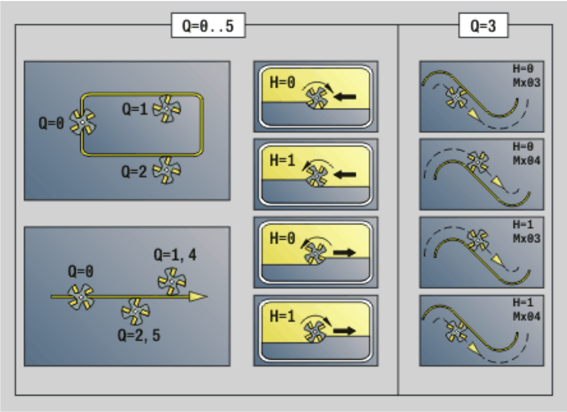
7 Returns to retraction plane RB.

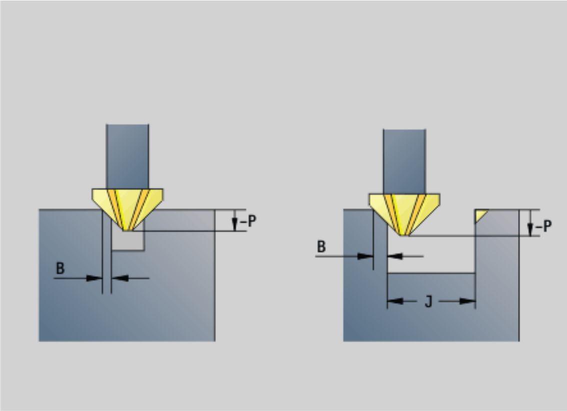
You can change the **machining direction and the cutter radius compensation** (TRC) with the **`cycle type Q`**, the **`cutting direction H`** and the rotational direction of the tool (see following table). Program only the parameters given in the following table.

Contour milling G840									
Cycle type	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Version**	**Cycle type**	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Description**
Contour (Q=0)	–	Mx03	–		Outside	Up-cut milling (H=0)	Mx04	Left	
Contour	–	Mx03	–		Outside	Climb milling (H=1)	Mx03	Left	
Contour	–	Mx04	–		Outside	Climb milling (H=1)	Mx04	Right	
Contour	–	Mx04	–		Contour (Q=0)	–	Mx03	–	
Inside (Q=1)	Up-cut milling (H=0)	Mx03	Right		Contour	–	Mx04	–	
Inside	Up-cut milling (H=0)	Mx04	Left		Right (Q=3)	Up-cut milling (H=0)	Mx03	Right	
Inside	Climb milling (H=1)	Mx03	Left		Left (Q=3)	Up-cut milling (H=0)	Mx04	Left	
Inside	Climb milling (H=1)	Mx04	Right		Left (Q=3)	Climb milling (H=1)	Mx03	Left	
Outside (Q=2)	Up-cut milling (H=0)	Mx03	Right		Right (Q=3)	Climb milling (H=1)	Mx04	Right	

G840—Deburring

G840 deburrs when you program **chamfer width B**. If there is any overlapping of the contour, specify with **cycle type Q** whether the first section (as of starting point) or the entire contour is to be machined. Program only the parameters given in the following table.





Parameters—Deburring

Q Cycle type (= milling location).

- Open contour. If there is any overlapping, Q defines whether the first section (as of starting point) or the entire contour is to be machined.
  - Q=0: Center of milling cutter on the contour (without TRC)
  - Q=1: Machining at the left of the contour. If there is any overlapping, G840 machines only the first section of the contour (starting point: 1st point of intersection).
  - Q=2: Machining at the right of the contour. If there is any overlapping, G840 machines only the first section of the contour (starting point: 1st point of intersection).
  - Q=3: The contour is machined to the left or right depending on H and the direction of cutter rotation (see table). If there is any overlapping, G840 machines only the first section of the contour (starting point: 1st point of intersection).
  - Q=4: Machining at the left of the contour. If there is overlapping, G840 machines the entire contour.
  - Q=5: Machining at the right of the contour. If there is overlapping, G840 machines the entire contour.
- Closed contours
  - Q=0: Center of milling cutter on the contour (hole position = starting point)
  - Q=1: Inside milling
  - Q=2: Outside milling
  - Q=3 to 5: Not allowed

ID Milling contour—name of the contour to be milled

NS Block number—beginning of contour section

- Figures: Block number of the figure
- Free open or closed contour: First contour element (not starting point)

NE Block number—end of contour section

- Figures, free closed contour: No input
- Free open contour: Last contour element
- Contour consists of one element:
  - No input: Machining in contour direction
  - NS=NE programmed: Machining against the contour direction

E Reduced feed rate for circular elements (default: current feed rate)

Parameters—Deburring

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Contour element is approached/departed tangentially on a line

P Milling depth (indicated as a negative value)

XS Milling top edge—lateral surface (replaces the reference plane from the contour definition)

ZS Milling top edge—face (replaces the reference plane from the contour definition)

RB Retraction plane (default: back to starting position)

- Front or read face: Return position in Z direction
- Lateral surface: Return position in X direction (diameter)

B Chamfer width when deburring the edges

J Preparation diameter. For open contours, the contour to be deburred is calculated from the programmed contour and J.

Remember that:

- J programmed: The cycle deburrs both sides of the slot (see 1 in the illustration).
- J not programmed: The deburring tool is so wide that both sides of the slot are deburred in one pass (see 2 in the illustration).

D Starting element number when partial figures are machined.

V End element number when partial figures are machined.

The direction of contour definition for figures is counterclockwise. The first contour element for figures:

- Circular slot: The larger arc
- Full circle: The upper semicircle
- Rectangles, polygons and linear slots: The orientation angle points to the first contour element.

A Sequence for "Milling, deburring": A=0 (default=0)

Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For figures, use D and V to select the approach/departure element.

Cycle run for deburring

1. Starting position (X, Z, C) is the position before the cycle begins.
2. Moves to the safety clearance and infeed to the first milling depth.
3. - J not programmed: Mills the programmed contour.
   - J programmed, open contour: Calculates and mills the "new" contour.
4. Returns to retraction plane RB.

Pocket milling, roughing G845

G840—Fundamentals

G845 roughs closed contours. Choose one of the following **plunge strategies**, depending on the milling cutter you are using:

- Plunge vertically
- Plunge at a pre-drilled position
- Plunge in a reciprocating or helical motion

When "plunging at a pre-drilled position," you have the following alternatives:

- **Calculate positions, drill, mill.** The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with "G845 A1 ..."
  - Predrill with "G71 NF .."
  - Call cycle "G845 A0 ..." The cycle positions the tool above the hole; the tool plunges and mills the pocket.
- **Drill, mill.** The machining process is performed in the following steps:
  - Drill a hole inside the pocket with "G71 ..."
  - Position the milling cutter above the hole and call "G845 A0 ..." The tool plunges and mills the section.

If the pocket consists of multiple sections, G845 takes all the sections of the pocket into account for drilling and milling. Call "G845 A0 .." separately for each section when calculating the hole positions without "G845 A1 ..".

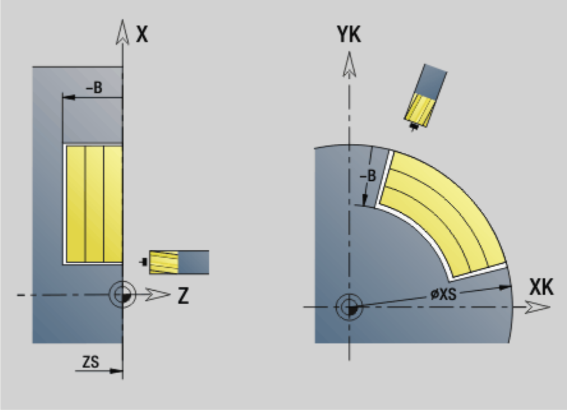
G845 takes the following oversizes into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

Program oversizes for calculating the hole positions **and** for milling.

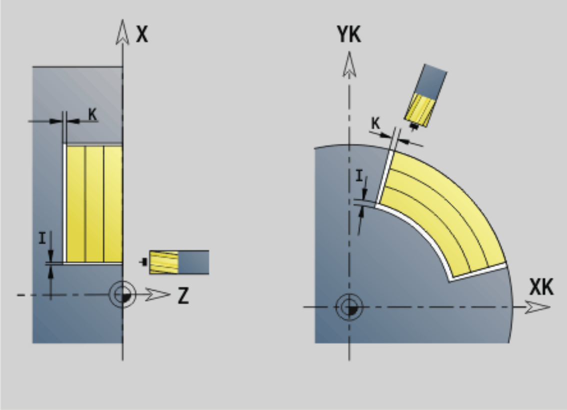
G845—Calculating hole positions

"G845 A1 .." calculates the hole positions and stores them at the reference specified in "NF." The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling "G845 A1 ..". Program only the parameters given in the following table.



See also:

- G845—Fundamentals: Page 358
- G845—Milling: Page 360



Parameters—Calculating hole positions

ID Milling contour—name of the contour to be milled

NS Starting block no. of contour

- Figures: Block number of the figure
- Free closed contour: A contour element (not starting point)

B Milling depth (default: depth from the contour description)

XS Milling top edge—lateral surface (replaces the reference plane from the contour definition)

ZS Milling top edge—face (replaces the reference plane from the contour definition)

I Oversize in X direction (radius)

K Oversize in Z direction

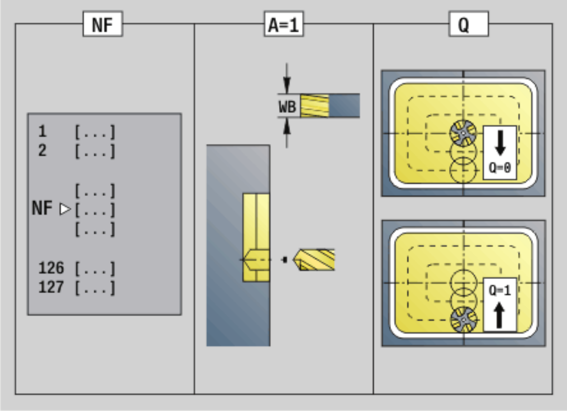
Q Machining direction (default: 0)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

A Sequence for "Calculate hole positions": A=1

NF Position mark—reference at which the cycle stores the hole positions [1 to 127].

WB Plunge length—diameter of the milling cutter



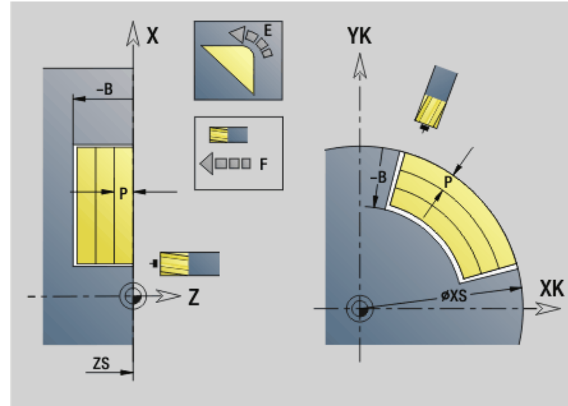
- G845 overwrites any hole positions that may still be stored at the reference "NF."
- The parameter "WB" is used both for calculating the hole positions and for milling. When calculating the hole positions, "WB" describes the diameter of the milling cutter.

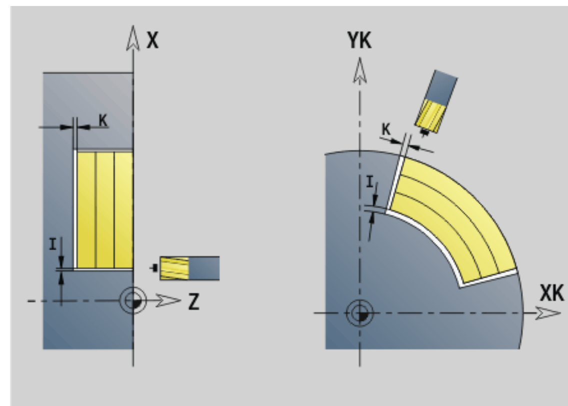
G845—Milling

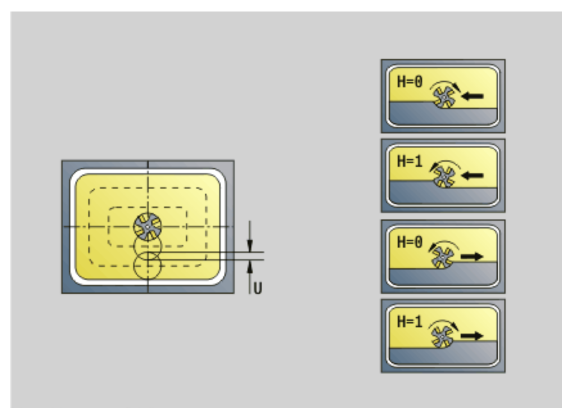
You can change the cutting direction with the **cutting direction H**, the **machining direction Q** and the direction of tool rotation (see following table). Program only the parameters given in the following table.

See also:

- G845—Fundamentals: Page 358
- G845—Calculating hole positions: Page 359







Parameters—Milling

ID Milling contour—name of the contour to be milled

NS Starting block no. of contour

- Figures: Block number of the figure
- Free closed contour: A contour element (not starting point)

B Milling depth (default: depth from the contour description)

P (Maximum) infeed (default: milling in one infeed)

XS Milling top edge—lateral surface (replaces the reference plane from the contour definition)

ZS Milling top edge—face (replaces the reference plane from the contour definition)

I Oversize in X direction (radius)

K Oversize in Z direction

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

V Overrun factor (no effect with C-axis machining)

H Cutting direction (default: 0)

- 0: Up-cut milling
- 1: Climb milling

F Feed rate for infeed (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

RB Retraction plane (default: back to starting position)

- Front or read face: Return position in Z direction
- Lateral surface: Return position in X direction (diameter)

Q Machining direction (default: 0)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

Parameters—Milling

A Sequence for "Milling": A=0 (default=0)

NF Position mark—reference from which the cycle reads the hole positions [1 to 127].

O Plunging behavior (default: 0)

O=0 (vertical plunge): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the pocket.

O=1 (plunge at pre-drilled position):

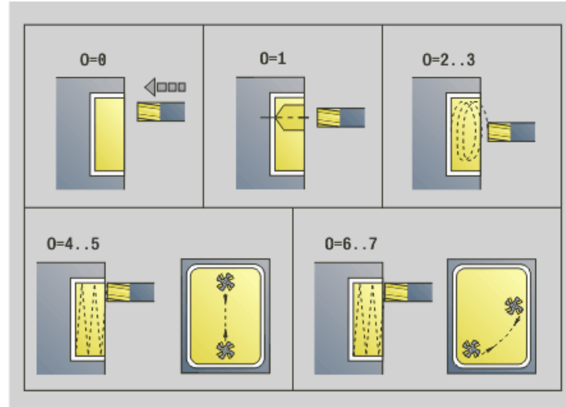
- If "NF" is programmed: The cycle positions the milling cutter above the first pre-drilled hole; the tool plunges and mills the first area. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next area, etc.
- If "NF" is not programmed: The tool plunges at the current position and mills the area. If applicable, position the tool to the next pre-drilled hole and mill the next area, etc.

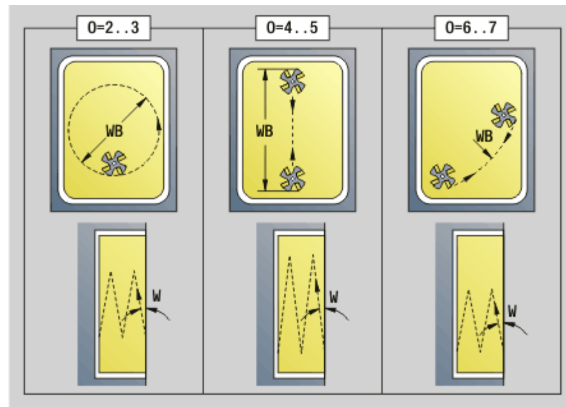
O=2, 3 (helical plunge): The tool plunges at the angle "W" and mills full circles with the diameter "WB." As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=2—manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=3—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas.

O=4, 5 (reciprocating linear plunge): The tool plunges at the angle "W" and mills a linear path of the length "WB." You can define the orientation angle in "WE." The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=4—manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=5—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas. The plunging position is determined from the type of figure and from "Q" as follows:





Parameters—Milling

- Q0 (from the inside toward the outside):
  – Linear slot, rectangle, polygon: Reference point of the figure
  – Circle: Circle center
  – Circular slot, "free" contour: Starting point of the innermost milling path
- Q1 (from the outside toward the inside):
  – Linear slot: Starting point of the slot
  – Circular slot, circle: Not machined
  – Rectangle, polygon: Starting point of the first linear element
  – "Free" contour: Starting point of the first linear element (at least one linear element must exist)

O=6, 7 (reciprocating circular plunge): The tool plunges at the plunging angle "W" and mills a circular arc of 90°. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling. "WE" defines the arc center, "WB" the arc radius.

- O=6—manually: The tool position corresponds to the center of the circular arc. The tool moves to the arc starting point and plunges.
- O=7—automatically (only permitted for circular slots and circles): The cycle calculates the plunging position on the basis of "Q:"
  - Q0 (from the inside toward the outside):
    – Circular slot: The circular arc lies on the curvature radius of the slot
    – Circle: Not permitted
  - Q1 (from the outside toward the inside): Circular slot, circle: The circular arc lies on the outermost milling path

W Plunging angle in infeed direction

WE Orientation angle of the milling path/circular arc. Reference axis:

- Front or rear face: Positive XK axis
- Lateral surface: Positive Z axis

Default orientation angle, depending on "O:"

- O=4: WE= 0°
- O=5 and
  - Linear slot, rectangle, polygon: WE= position angle of the figure
  - Circular slot, circle: WE=0°
  - "Free" contour and Q0 (from the inside toward the outside): WE=0°
  - "Free" contour and Q1 (from the outside toward the inside): Orientation angle of the starting element

WB Plunge length/plunge diameter (default: 1.5 * milling diameter)

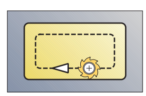
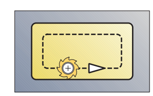
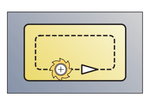
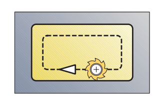
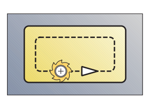
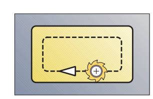
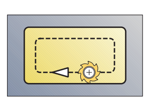
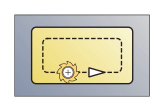
For the machining direction Q=1 (from the outside toward the inside), please note:

- The contour must start with a linear element.
- If the starting element is < WB, WB is reduced to the length of the starting element.
- The length of the starting element must not be less than 1.5 times the diameter of the milling cutter.

Cycle run

1 Starting position (X, Z, C) is the position before the cycle begins.
2 Calculates the number of cuts (infeeds to the milling planes, infeeds in the milling depths) and the plunging positions and paths for reciprocating or helical plunges.
3 Approaches to safety clearance and, depending on O, feeds to the first milling depth or approaches helically or on a reciprocating path.
4 Mills a plane.
5 Retracts by the safety clearance, returns and cuts to the next milling depth.
6 Repeat steps 4 and 5 until the complete surface is milled.
7 Returns to retraction plane RB.

You can change the **milling direction** with the cutting direction H, the machining direction Q and the direction of tool rotation (see following table). Program only the parameters given in the following table.

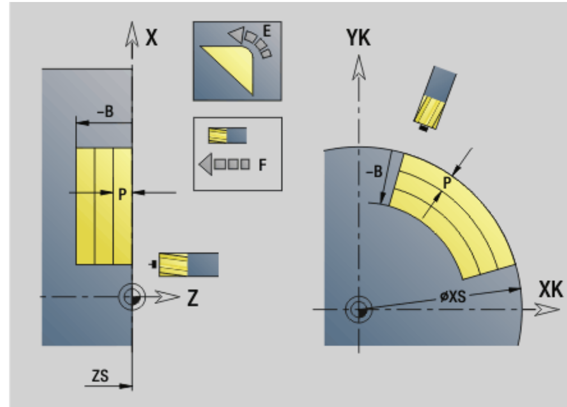
Pocket milling, roughing G845							
Cutting direction	**Machining direction**	**Direction of tool rotation**	**Execution**	**Cutting direction**	**Machining direction**	**Direction of tool rotation**	**Description**
Up-cut milling (H=0)	From inside (Q=0)	Mx03		Climb milling (H=1)	From inside (Q=0)	Mx03	
Up-cut milling (H=0)	From inside (Q=0)	Mx04		Climb milling (H=1)	From inside (Q=0)	Mx04	
Up-cut milling (H=0)	From outside (Q=1)	Mx03		Climb milling (H=1)	From outside (Q=1)	Mx03	
Up-cut milling (H=0)	From outside (Q=1)	Mx04		Climb milling (H=1)	From outside (Q=1)	Mx04	

Pocket milling, finishing G846

G846 finish-machines closed contours.

If the pocket consists of multiple sections, G846 takes all the sections of the pocket into account.

You can change the **cutting direction** with the **`cutting direction H`**, the **`machining direction Q`** and the direction of tool rotation (see following table).



Parameters—finishing

ID Milling contour—name of the contour to be milled

NS Starting block no. of contour

- Figures: Block number of the figure
- Free closed contour: A contour element (not starting point)

B Milling depth (default: depth from the contour description)

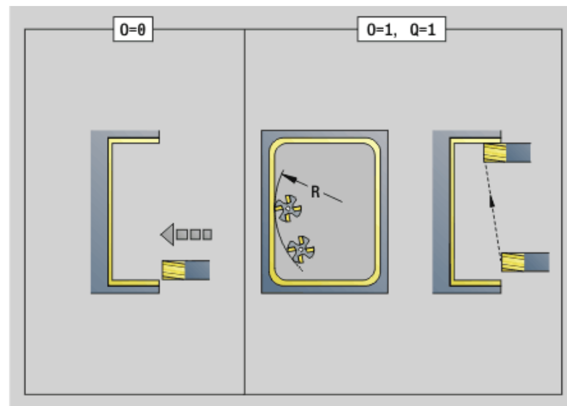
P (Maximum) infeed (default: milling in one infeed)

XS Milling top edge—lateral surface (replaces the reference plane from the contour definition)

ZS Milling top edge—face (replaces the reference plane from the contour definition)

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly. Feed to the starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.



U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

V Overrun factor—no effect with C-axis machining

H Cutting direction (default: 0)

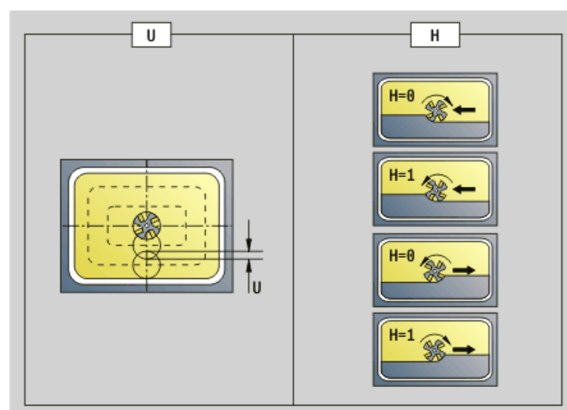
- 0: Up-cut milling
- 1: Climb milling

F Feed rate for infeed (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

RB Retraction plane (default: back to starting position)

- Front or read face: Return position in Z direction
- Lateral surface: Return position in X direction (diameter)



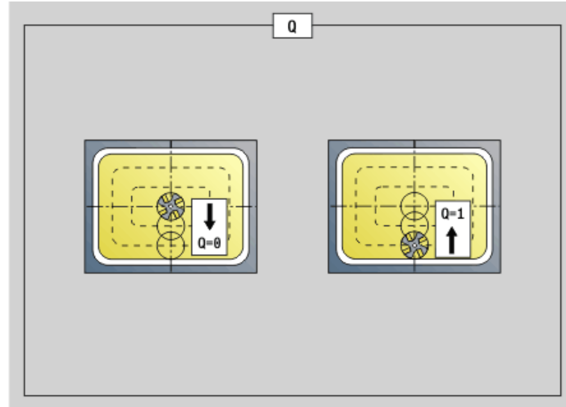
Parameters—finishing

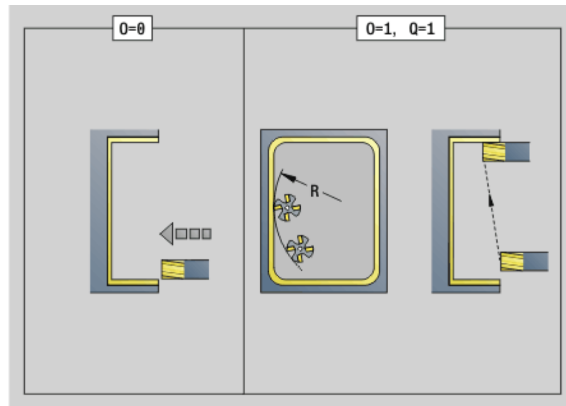
Q Machining direction (default: 0)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

O Plunging behavior (default: 0)

- O=0 (vertical plunge): The cycle moves the tool to the starting point; the tool plunges and finishes the pocket.
- Q=1 (approaching arc with depth feed): When machining the upper milling planes, the tool advances to the milling plane and then approaches on an arc. When machining the bottom milling plane, the tool plunges to the milling depth while moving on the approaching arc (three-dimensional approaching arc). You can use this approach behavior only in conjunction with an approaching arc "R" and when machining from the outside toward the inside (Q=1).

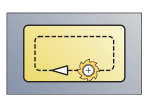
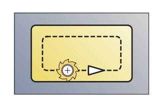
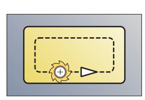
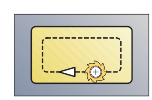




Cycle run

1 Starting position (X, Z, C) is the position before the cycle begins.
2 Calculates the number of cutting passes (infeeds to the milling planes, infeeds in the milling depths).
3 Moves to the safety clearance and feeds to the first milling depth.
4 Mills a plane.
5 Retracts by the safety clearance, returns and cuts to the next milling depth.
6 Repeat steps 4 and 5 until the complete surface is milled.
7 Returns to retraction plane RB.

You can change the **cutting direction** with the **`cutting direction H`**, the **`machining direction Q`** and the direction of tool rotation (see following table).

Pocket milling, finishing G846					
Cutting direction	**Direction of tool rotation**	**Execution**	**Cutting direction**	**Direction of tool rotation**	**Execution**
Up-cut milling (H=0)	Mx03		Climb milling (H=1)	Mx03	
Up-cut milling (H=0)	Mx04		Climb milling (H=1)	Mx04	

4.27 Engraving cycles

Character set

The Control can realize the characters listed in the following table. The text to be engraved is entered as a character string. Diacritics and special characters that you cannot enter in the editor can be defined, character by character, in NF. If text is defined in "ID" and a character is defined in "NF," the text is engraved before the character.

Small letters		**Capital letters**		**Numerals, diacritics**		**Special characters**		
NF	**Character**	**NF**	**Character**	**NF**	**Character**	**NF**	**Character**	**Meaning**
97	a	65	A	48	0	32		Blank space
98	b	66	B	49	1	37	%	Per cent sign
99	c	67	C	50	2	40	(	Opening parenthesis
100	d	68	D	51	3	41	)	Closing parenthesis
101	e	69	E	52	4	43	+	Plus character
102	f	70	F	53	5	44	,	Comma
103	g	71	G	54	6	45	–	Minus sign
104	h	72	H	55	7	46	.	Point
105	i	73	I	56	8	47	/	Forward slash
106	j	74	J	57	9	58	:	Colon
107	k	75	K			60	<	Less than character
108	l	76	L	196	Ä	61	=	Equal sign
109	m	77	M	214	Ö	62	>	Greater than character
110	n	78	N	220	Ü	64	@	at
111	o	79	O	223	ß	91	[	Opening brackets
112	p	80	P	228	ä	93	]	Closing brackets
113	q	81	Q	246	ö	95	_	Underscore
114	r	82	R	252	ü	8364		Euro sign
115	s	83	S			181	µ	Micro
116	t	84	T			186	°	Degrees
117	u	85	U			215	*	Multiplication sign
118	v	86	V			33	!	Exclamation point

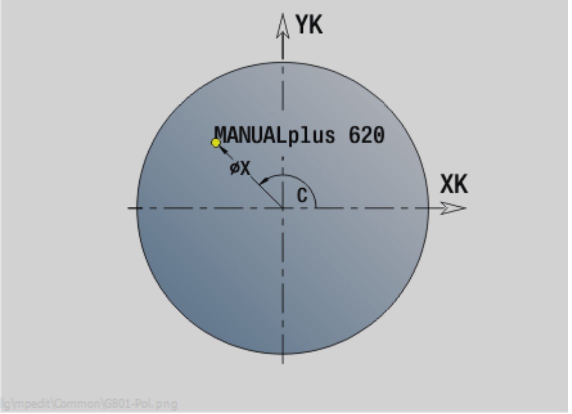
Small letters		Capital letters		Numerals, diacritics		Special characters		
NF	Character	NF	Character	NF	Character	NF	Character	Meaning
119	w	87	W			38	&	Ampersand and
120	x	88	X			63	?	Question mark
121	y	89	Y			174	®	Trademark
122	z	90	Z			216	Ø	Diameter sign

Engraving on front face G801

G801 engraves character strings in linear or polar layout on the front face. For character table and more information, see page 366

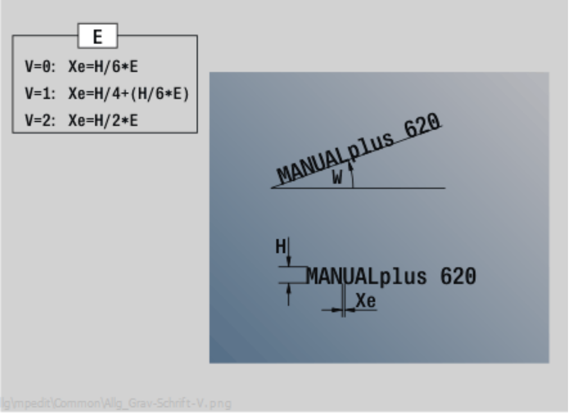
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

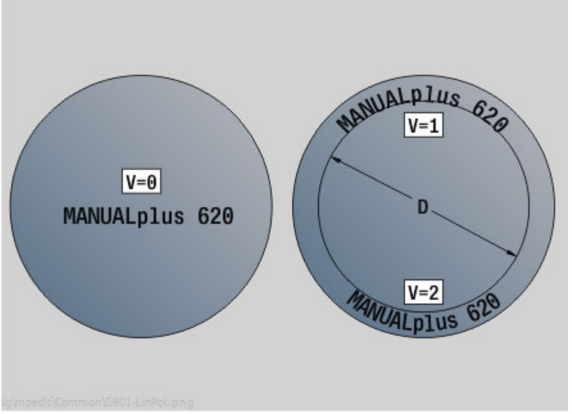
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.



Parameters

X, C	Polar starting point
XK, YK	Cartesian starting point
Z	End point. Z position, infeed depth during milling.
RB	Retraction plane. Z position retracted to for positioning.
ID	Text to be engraved
NF	Character number (character to be engraved)
W	Inclination angle. Example: 0° = Vertical characters: the characters are aligned in sequence in positive X direction
H	Font height
E	Distance factor (for calculation see figure)
V	Execution ■ 0: Linear ■ 1: Arched above ■ 2: Arched below
D	Reference diameter
F	Plunging feed rate factor (plunging feed rate = current feed rate * F)





Engraving on lateral surface G802

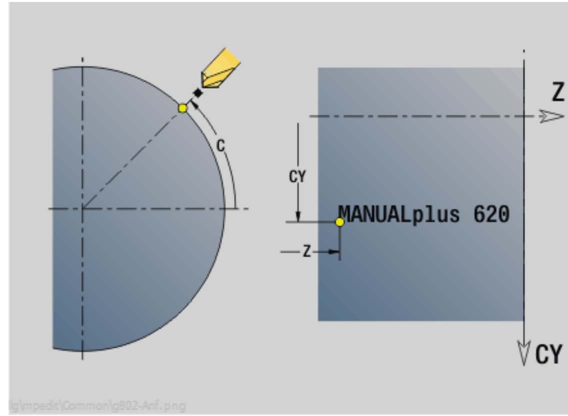
G802 engraves character strings aligned linearly on the lateral surface. For character table and more information, see page 366

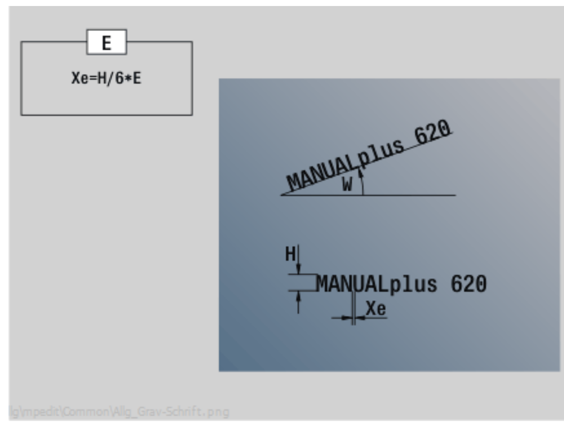
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

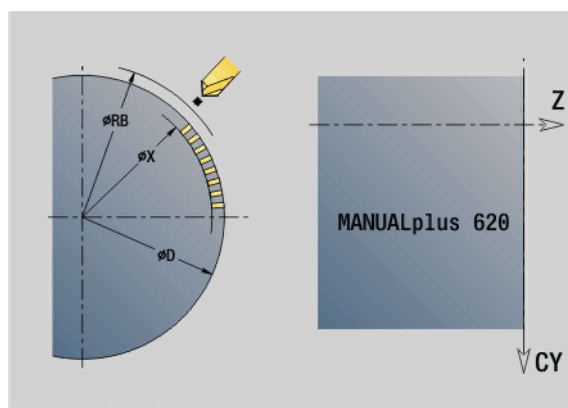
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Parameters

Z	Start point
C	Starting angle
CY	Start point
X	Final point (diameter). X position, infeed depth during milling.
RB	Retraction plane. X position retracted to for positioning.
ID	Text to be engraved
NF	Character number. ASCII code of the character to be engraved
W	Inclination angle
H	Font height
E	Distance factor (for calculation see figure)
D	Reference diameter
F	Plunging feed rate factor (plunging feed rate = current feed rate * F)







4.28 Contour follow-up

Automatic contour follow-up is not possible with program branches or repetitions. In these cases you control the contour follow up with the following commands.

Saving/loading contour follow-up G702

G702 saves the current contour or loads a saved contour.

Parameters

ID Workpiece blank contour—name of the auxiliary workpiece blank

Q Save/load contour

- 0: Saves the current contour. The contour follow-up is not affected.
- 1: Loads the specified contour. The contour follow-up is continued with the loaded contour.
- 2: The following cycle uses the "internal workpiece blank."

H Memory number (0 .. 9)

V The following information is saved:

- 0: Everything (variable contents and workpiece blank contours)
- 1: Variable contents
- 2: Workpiece blank contours

G702 Q=2 switches off the global contour follow-up for the following cycle. Once the cycle has been executed, the global contour follow-up is effective again.

The affected cycle uses the "internal workpiece blank." The cycle determines the internal workpiece blank from the contour and the tool position.

G702 Q2 must be programmed before the cycle.

Contour follow-up on/off G703

G703 is used to deactivate/reactivate the contour follow-up.

Parameter

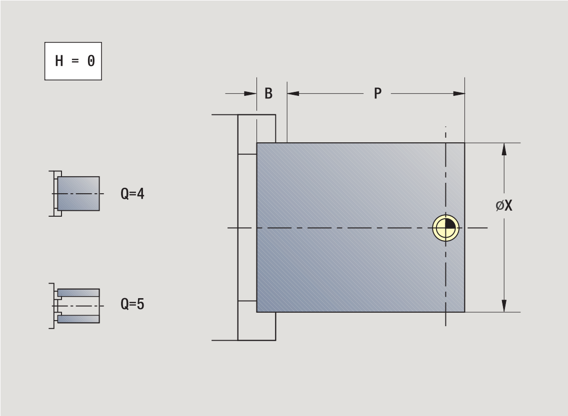
Q Contour follow-up on/off

- 0: Off
- 1: On

4.29 Other G functions

Chucking equipment in simulation G65

G65 displays the selected chucking equipment in the simulation graphics.



Parameters

H No. of clamping (no. of chuck) (always program H=0)
X Diameter of workpiece blank
Z Start point—No input
D No. of spindle—No input:
Q Chuck form

- 4: Externally clamped
- 5: Internally clamped

B Clamping length (B+P = length of blank)
P Unclamping length (free length)

Workpiece blank contour G67 (for graphics)

G67 displays an auxiliary workpiece blank in the simulation graphics.

Parameters

ID ID of auxiliary workpiece blank
NS Block number of contour

Period of dwell G4

With G4, the Control interrupts the program run for the time F before executing the next program block. If G4 is programmed together with a path of traverse in the same block, the dwell time only becomes effective after the path of traverse has been executed.

Parameter

F Dwell time [sec] (0 < F <= 999)

Precision stop G7

G7 switches precision stop on. It is a modal function. With a precision stop, the Control does not run the following block until the last point has been reached in the tolerance window for position. The tolerance window is a configuration parameter ("ParameterSets PX(PZ)/CfgControllerTol/posTolerance").

Precision stop affects single contours and cycles. The NC block containing G7 is also executed with a precision stop.

Precision stop off G8

G8 switches precision stop off. The block containing G8 is executed **without** a precision stop.

Precision stop G9

G9 activates a precision stop for the block in which it is programmed. With a precision stop, the Control does not run the following block until the last point has been reached in the tolerance window for position. The tolerance window is a configuration parameter ("ParameterSets PX / PZ > CfgControllerTol > posTolerance").

Switch off protection zone G60

G60 is used to cancel protection zone monitoring. G60 is programmed **before** the traversing command to be monitored or not monitored.

Parameter

Q Activate/Deactivate

- 0: Activate protection zone (modal)
- 1: Deactivate protection zone (modal)

Application example: With G60, you can temporarily deactivate a programmed monitoring of the protection zone in order to machine a centric through hole.

Example: G60

```
. . .
N1 T4 G97 S1000 G95 F0.3 M3
N2 G0 X0 Z5
N3 G60 Q1 [Deactivate the protection zone]
N4 G71 Z-60 K65
N5 G60 Q0 [Activate the protection zone]
. . .
```

Actual values in variables G901

G901 transfers the actual values of all the axes of a slide into the variables for the interpolation information.

See G904 Page 373.

Zero-point shift in variables G902

G902 transfers the zero-point shifts into the variables for the interpolation information.

See G904 Page 373.

Lag error in variables G903

G903 transfers the current following error (distance by which the actual values lags the nominal value) into the variables for the interpolation information.

See G904 Page 373.

Read interpolation information G904

G904 transfers all the current interpolation information on the current slide to the variable memory.

Interpolation information	
#a0(Z,1)	Zero-point shift of the Z axis of slide $1
#a1(Z,1)	Actual position of the Z axis of slide $1
#a2(Z,1)	Nominal position of the Z axis of slide $1
#a3(Z,1)	Lag error of the Z axis of slide $1
#a4(Z,1)	Distance to go in the Z axis of slide $1
#a5(Z,1)	Logical axis number of the Z axis of slide $1
#a5(0,1)	Logical axis number of the main spindle
#a6(0,1)	Rotational direction of main spindle $1
#a9(Z,1)	Trigger position of the touch probe
#a10(Z,1)	IPO axis value

Interpolation information syntax	
Syntax:	**#an(axis,channel)**
	■ n = number of the information ■ axis = name of the axis ■ channel = slide number

Feed rate override 100% G908

G908 sets the feed override for traverse paths (G0, G1, G2, G3, G12, G13) block by block to 100%.

Program G908 and the traverse path in the same NC block.

Interpreter stop G909

The Control pre-interprets the NC blocks. If variables are assigned shortly before the evaluation, "old values" would be processed. G909 stops the pre-interpretation. The NC blocks are processed up to G909. Only after G909, are the subsequent NC blocks processed.

Apart from G909, the NC block should only contain synchronous functions. (Some G functions generate an interpreter stop.)

Spindle override 100% G919

G919 is used to deactivate/activate the spindle speed override.

Parameters

Q Spindle number (default: 0)

H Type of limit (default: 0)

- 0: Activate spindle speed override
- 1: Spindle override at 100%—modal
- 2: Spindle override at 100%—for the current NC block

Deactivate zero-point shifts G920

G920 deactivates the workpiece zero point and zero-point shifts. Traverse paths and position values are referenced to the **distance tool tip – machine zero point.**

Deactivate zero-point shifts, tool lengths G921

G921 deactivates the workpiece zero point, zero-point shifts and tool dimensions. Traverse paths and position values are referenced to the **slide reference point – machine zero point.**

End position of tool G922

With G922 you can position the active tool to a defined angle.

Parameter

C Angular position for tool orientation

Fluctuating spindle speed G924

To reduce resonant vibrations you can use G924 to program a changing spindle speed. In G924 you define the time interval and the range for the speed change. The G924 function is automatically reset at the end of the program. You can also deactivate the function through another call with the setting H=0 (off).

Parameters

Q Spindle number (machine-dependent)

K Repetition rate: Time interval in hertz (repetitions per second)

I Change of spindle speed

H Switch on/off the G924 function

- 0: Off
- 1: On

Convert lengths G927

Function G927 is used to convert the tool lengths at the current tool insert angle to the initial position of the tool (reference position in B axis = 0).

The results can be interrogated in the variables #n927(X), #n927(Z), and #n927(Y).

Parameters

H Method of conversion:

- 0: Convert tool length to reference position (take I + K of the tool into account)
- 1: Convert tool length to reference position (do not take I + K of the tool into account)
- 2: Convert tool length from the reference position to the current work position (take I + K of the tool into account)
- 3: Convert tool length from the reference position to the current work position (do not take I + K of the tool into account)

X, Y, Z Axis values (X value = radius). If nothing is entered, the value 0 is used.

Calculate variables automatically G940

Use G940 to convert metric values to inch values. When you create a new program you can select between **`metric`** units and **`inches`**. Internally the control always calculates with metric values. If you read out variables in an "inch" program, the variables are always output as metric values. Use G940 to convert the variables to INCH values.

Parameter

H Switch on/off the G940 function

- 0: Unit conversion active
- 1: Units remain metric

In inch programs, a conversion is required for variables that refer to a metric unit of measurement:

Machine dimensions	
#m1(n)	Machine dimensions of an axis, e.g. #m1(X) for machine dimensions of the X axis

Tool data, reading...	
#wn(NL)	Usable length (inside turning and drilling tools)
#wn(RS)	Cutting radius
#wn(ZD)	Stud diameter
#wn(DF)	Cutter diameter
#wn(SD)	Shank diameter

Tool data, reading...	
#wn(SB)	Cutting width
#wn(AL)	Length of first cut
#wn(FB)	Cutter width
#wn(ZL)	Tool setting dimension in Z
#wn(XL)	Tool setting dimension in X
#wn(YL)	Tool setting dimension in Y
#wn(I)	Position of tool tip center in X
#wn(K)	Position of tool tip center in Z
#wn(ZE)	Distance between tool tip and slide zero point Z
#wn(XE)	Distance between tool tip and slide zero point X
#wn(YE)	Distance between tool tip and slide zero point Y

Reading the current NC information	
#n0(Z)	Last programmed position Z
#n120(X)	Reference diameter X for calculating CY
#n57(X)	Oversize in X
#n57(Z)	Oversize in Z
#n58(P)	Equidistant oversize
#n150(X)	Cutting width shifted in X by G150
#n95(F)	Last programmed feed rate
#n47(P)	Current safety clearance
#n147(I)	Current safety clearance in working plane
#n147(K)	Current safety clearance in infeed direction

Internal information for defining constants	
__n0_x	768 Last programmed position X
__n0_y	769 Last programmed position Y
__n0_z	770 Last programmed position Z
__n120_x	787 Reference diameter X for calculating CY
__n57_x	791 Oversize in X
__n57_z	792 Oversize in Z

Internal information for defining constants	
__n58_p	793 Equidistant oversize
__n150_x	794 Cutting width shifted in X by G150/G151
__n150_z	795 Cutting width shifted in Z by G150/G151
__n95_f	800 Last programmed feed rate

Read interpolation information G904	
#a0(Z,1)	Zero-point shift of the Z axis of slide $1
#a1(Z,1)	Actual position of the Z axis of slide $1
#a2(Z,1)	Nominal position of the Z axis of slide $1
#a3(Z,1)	Lag error of the Z axis of slide $1
#a4(Z,1)	Distance to go in the Z axis of slide $1

Misalignment compensation G976

With the G976 function (misalignment compensation) you can run the following operations on tapering contours (e.g. to counter a mechanical offset). The G976 function is automatically reset at the end of the program. You can also deactivate the function through another call with the setting H=0 (off).

Parameters

Z Starting point
K Length
I Incremental distance
J Incremental distance
H Switch on/off the G976 function

- 0: Off
- 1: On

Activating zero-point shifts G980

G980 activates the workpiece zero point and all zero-point shifts. Traverse paths and position values are referenced to the distance of the **tool tip to the workpiece zero point,** while taking the zero point shifts into consideration.

Activate zero-point shifts, tool lengths G981

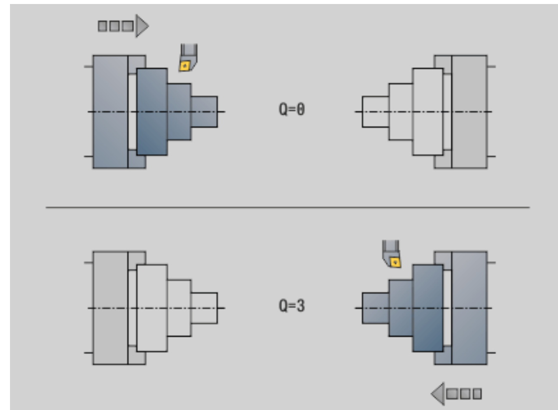
G981 activates the workpiece zero point, all zero-point shifts and the tool dimensions. Traverse paths and position values are referenced to the distance of the **tool tip to the workpiece zero point,** while taking the zero point shifts into consideration.

Activate direct program-run continuation G999

With the G999 function, when running a program in Single Block mode, the following NC blocks are run with a single NC start to the end of the program. G999 is then deactivated by again calling the function with the setting Q=0 (off).

Converting and mirroring G30

The G30 function converts G functions, M functions and spindle numbers. G30 mirrors traverse paths and tool dimensions and shifts the machine zero point about the "zero point offset" of the axis (machine parameter: Trans_Z1).



Parameters

H Table number of the conversion table (possible only if the machine tool builder has configured a conversion table).

Q Spindle number

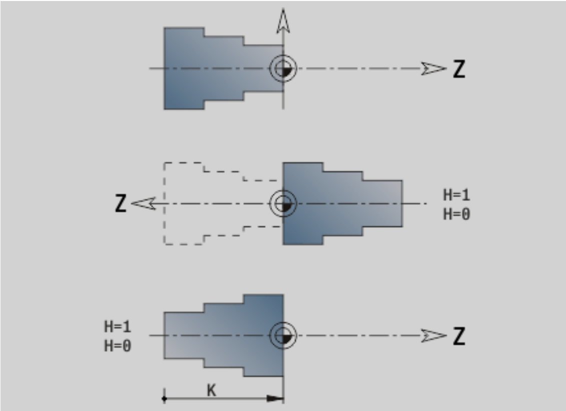
Application: For full-surface machining, you describe the complete contour, machine the front face, rechuck the workpiece using an expert program, and then machine the rear face. To enable you to program rear-face machining in the same way as front-face machining (Z axis orientation, arc rotational direction, etc.). Includes the expert program commands for converting and mirroring.

Danger of collision!

- In the transition from AUTOMATIC to MANUAL OPERATION, conversions and mirror images are retained
- Switch off the conversion/mirroring if you activate the front-face machining after rear-face machining (for example during program section repeats with M99)
- After a new program selection, the conversion/mirroring is switched off (example: transition from MANUAL to AUTOMATIC mode)

Transformations of contours G99

With the G99 function you can mirror contours, shift them and bring the workpiece to the desired machining position.



Parameters

Q	Function is not yet supported.
D	Spindle number
X	Shift in X (diameter value)
Z	Shift in Z
V	Mirroring the Z axis of the coordinate system ■ Q=0: Do not mirror ■ Q=1: Mirror
H	Transformation type ■ H=0: Contour shift, not mirroring ■ H=1: Contour shift, mirroring and reversing the direction of the contour description
K	Shift length of tool: shift coordinate system in Z direction
O	Hide elements during transformation ■ O=0: All contours are transformed ■ O=1: Auxiliary contours are not transformed. ■ O=2: Face contours are not transformed ■ O=4: Lateral contours are not transformed You can also add input values in order to combine various settings (e.g. O=3 Do not transform auxiliary contours or face contours)

- Program G99 again if the workpiece is transferred to another spindle and/or moves its position in the working space.

Spindle synchronization G720

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

G720 controls the workpiece transfer from the master to the slave spindle and synchronizes functions such as polygonal turning jobs. The function stays active until you deactivate G720 with the setting H0.

If you would like to synchronize more than two spindles you can program G720 several times in succession.

Parameters

S	Number of the master spindle
H	Number of the slave spindle—no input or H=0: Switches off the spindle synchronization
C	Offset angle [°]
Q	Master spindle speed factor Range: -100 <= Q <= 100
F	Slave spindle speed factor Range: –100 <= F <= 100

Program the speed of the master spindle with Gx97 S.. and define the speed ratio between the master spindle and the slave spindle with Q, F. If you enter a negative value for Q or F, the direction of rotation of the slave spindle will be reversed.

Remember that: **Q * master speed = F * slave speed**

Example: G720

. . .	
N.. G397 S1500 M3	Speed of direction of rotation of master spindle
N.. G720 C180 S0 H1 Q2 F-1	Synchronization of master spindle and slave spindle. The slave spindle precedes the master spindle by 180°. Slave spindle: Direction of rotation M4; rotational speed 750
N.. G1 X.. Z..	
. . .	

C-angle offset G905

G905 measures the angular offset of workpiece transfer with rotating spindle. The sum of angle C and the angle offset goes into effect as the zero point shift of C axis. If you request the datum shift of the current C-axis in the variable #a0 (C,1) the sum of the programmed datum shift and the measured offset angle is transferred.

The zero offset is effective internally as a direct zero point shift for the respective C axis. The contents of the variables are retained even if the control has been switched off.

You can also examine and reset the respectively active datum shift of the C axis in the "Setup" menu in the "Set C-axis value" function.

Parameters

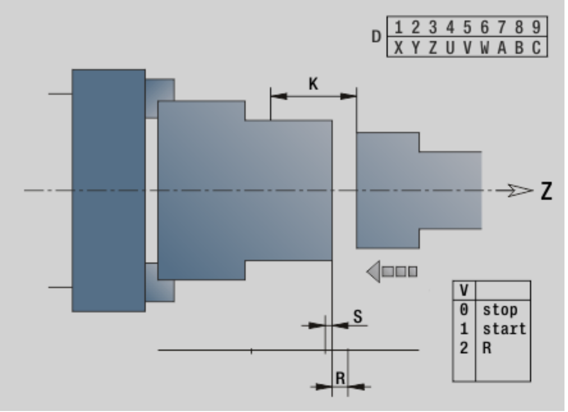
Q	Number of the C axis
C	Angle of additional zero point shift for offset gripping (–360° <= C <= 360°) – (default: 0°)

Danger of collision!

- For narrow workpieces the jaws have to grip at an offset.
- The zero point shift, C axis, remains in effect:
  - After switch from Automatic to Manual mode
  - After switch-off

Traversing to a fixed stop G916

The machine tool builder determines the scope of function and behavior of G916. The machine manual provides further information.



G916 switches on the "monitoring of the traverse path" and moves to a fixed stop (example: transferring a premachined workpiece to a second traveling spindle if you do not know the exact position of the workpiece).

The control stops the slide and saves the stop position. G916 generates an interpreter stop.

Parameters

H	Clamping force in daNewtons (1 daNewton = 10 newtons)
D	Axis number (X=1, Y=2, Z=3, U=4, V=5, W=6, A=7, B=8, C=9)
K	Incremental distance
R	Return path of traverse
V	Type of departure ■ V=0: Stay at fixed stop ■ V=1: Return to start position ■ V=2: Retract by return path **R**
O	Error evaluation ■ O=0: Error evaluation in expert program ■ O=1: The control issues an error message

Lag error monitoring is not activated until the acceleration phase has been completed.

The feed rate override is not effective during cycle execution.

Traversing to a fixed stop

When traversing to a fixed stop, the control moves

- up to the fixed stop and stops as soon as the following error has been reached. The remaining path of traverse is deleted
- back to starting position
- back by the return traverse path

Programming "traverse to a fixed stop":

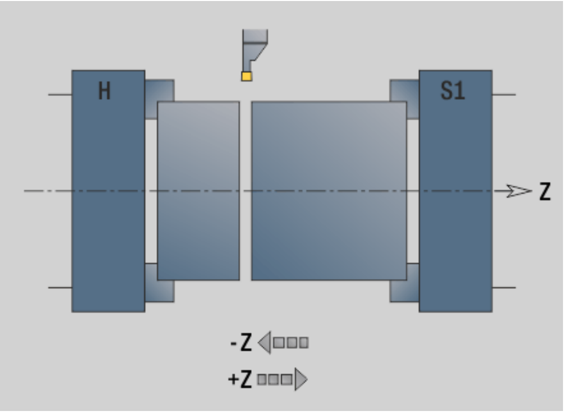
- Position the slide at a sufficient distance before the fixed stop.
- Use a moderate feed rate (< 1000 mm/min)

Example of traversing to a fixed stop:

. . .	
N.. G0 Z20	Pre-position slide 2
N.. G916 H100 D6 K-20 V0 O1	Activate monitoring, traverse to a fixed stop
. . .	

Controlled parting using lag error monitoring G917

The machine tool builder determines the scope of function and behavior of G917. The machine manual provides further information.



G917 "monitors" the path of traverse. The controlled parting function (cut-off control) prevents collisions caused by incomplete parting processes.

The control stops the slide when the tensile force is too high and generates an "interpreter stop."

Parameters

H	Tensile force
D	Axis number (X=1, Y=2, Z=3, U=4, V=5, W=6, A=7, B=8, C=9)
K	Incremental distance
O	Error evaluation ■ O=0: Error evaluation in expert program ■ O=1: The control issues an error message

During parting control, the parted workpiece moves in the positive Z direction. If a following error occurs, the workpiece is considered unparted.

The result is saved in the variable #i99:

- 0: Workpiece was not correctly cut off (following error detected)
- 1: Workpiece was correctly cut off (no following error detected)

Lag error monitoring is not activated until the acceleration phase has been completed.

The feed rate override is not effective during cycle execution.

Force reduction G925

The machine tool builder determines the scope of function and behavior of G925. The machine manual provides further information.

G925 activates/deactivates the force reduction. When the monitoring is activated, the maximum contact force for one axis is defined. Force reduction can be activated for only one axis per NC channel.

The G925 function limits the contact force for subsequent movements of the defined axis. G925 does not execute any traverse.

Parameters

H	Contact force [dN] – The contact force is limited to the given value
Q	Axis number (X=1, Y=2, Z=3, U=4, V=5, W=6, A=7, B=8, C=9)
S	Sleeve monitoring ■ 0: Deactivate (do not monitor the contact force) ■ 1: Activate (the contact force is monitored)

Lag error monitoring is not activated until the acceleration phase has been completed.

Sleeve monitoring G930

The machine tool builder determines the scope of function and behavior of G930. The machine manual provides further information.

G930 activates/deactivates the sleeve monitoring. When the monitoring is activated, the maximum contact force for one axis is defined. Sleeve monitoring can be activated for only one axis per NC channel.

G930 moves the defined axis by the programmed distance **D** until the defined contact force **H** has been reached.

Parameters

H	Contact force [dN] – The contact force is limited to the given value
Q	Axis number (X=1, Y=2, Z=3, U=4, V=5, W=6, A=7, B=8, C=9)
D	Incremental distance

Application example: G930 is applied to use the opposing spindle as a mechatronic tailstock. In this case the opposing spindle is equipped with a dead center and the contact force is limited with G930. A prerequisite for this application is a PLC program from the machine tool builder that enables the user to operate the mechatronic tailstock in the Manual and Automatic operating mode.

Lag error monitoring is not activated until the acceleration phase has been completed.

Tailstock function

With the tailstock function, the control moves up to the workpiece and stops as soon as the contact force has been reached. The remaining path of traverse is deleted.

Example of tailstock function

. . .	
N.. G0 Z20	Pre-position slide 2
N.. G930 H250 D6 K-20	Activate the tailstock function—contact force 250 daN
. . .	

4.30 Data input and data output

"WINDOW"—Output window for variables

WINDOW (x) opens an output window with x lines. The window is opened as a result of the first input/output. WINDOW (0) closes the window.

Syntax:
WINDOW(line number) (0 <= line number <= 20)

The standard window comprises 3 lines. You do not need to program it.

Example:

```
. . .
N  1 WINDOW(8)
N  2 INPUT("query: ",#l1)
N  3 #l2=17*#l1
N  4 PRINT("result: ",#l1,"*17 = ",#l2)
. . .
```

"WINDOW"—Output file for variables

The command WINDOW (x,"filename") saves the PRINT instruction in a file with the defined name and the extension **.LOG**, in the directory "V:\nc_prog\". The file is overwritten when the WINDOW command is run again.

Syntax:
WINDOW (line number, "filename")

Example:

```
. . .
N  1 WINDOW(8)
N  2 INPUT("query: ",#l1)
N  3 #l2=17*#l1
N  4 PRINT("result: ",#l1,"*17 = ",#l2)
. . .
```

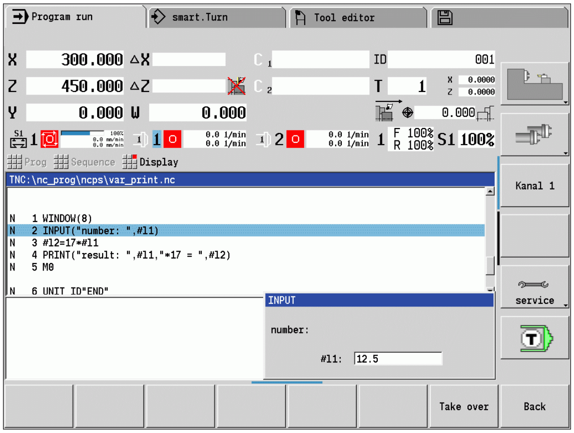
"INPUT"—Input of variables

Use INPUT to program the input of variables.

Syntax:
INPUT("text",variable)

You define the input text and the number of the variable. The Control stops the interpretation at INPUT, outputs the text and waits for input of the variable value.

The Control displays the input after having completed the INPUT command.



Output of # variables PRINT

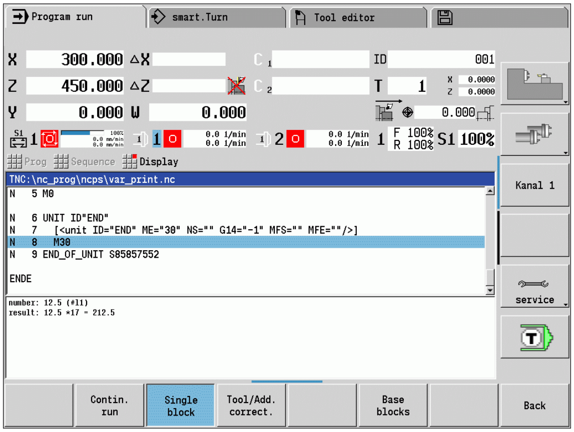
PRINT can be used to output texts and variable values during program run. You can program a succession of several texts and variables.

Syntax:

PRINT(text,variable, text,variable, ..)

Example:

PRINT("result: ",#l1,"*17 = ",#l2)



4.31 Programming variables

The Control provides a variety of variable types.

The following rules apply to the use of variables:

- Multiplication/division before addition/subtraction
- Up to 6 bracket levels
- **Integer variables:** Integer values between –32767 and +32768
- **Real variables:** Floating point numbers with max. 10 integers and 7 decimal places
- Do not use any blank spaces when programming variables.
- The variable number itself and an index value, if applicable, can be described by another variable, e.g.: #g(#c2)
- See the table for the available mathematical operations

- The distinction made by CNCPILOT XXXX and MANUALplus X110 controls between variables that can be modified at runtime and those that cannot, does not apply any longer. The NC program is no longer compiled before the program run, but at runtime.
- Program NC blocks containing variable calculations with "slide code $.." if your lathe has more than one slide. Otherwise, the calculations are repeated.
- Positions and dimensions transferred into system variables are always indicated in metric form. This also applies when an NC program is run in inches.

Syntax	Mathematical functions
+	Addition
–	Subtraction
*	Multiplication
/	Division
SQRT(...)	Square root
ABS(...)	Absolute amount
TAN(...)	Tangent (in degrees)
ATAN(...)	Arc tangent (in degrees)
SIN(...)	Sine (in degrees)
ASIN(...)	Arc sine (in degrees)
COS(...)	Cosine (in degrees)
ACOS(...)	Arc cosine (in degrees)
ROUND(...)	Round
LOGN(...)	Natural logarithm
EXP(...)	Exponential function ex
INT(...)	Truncate decimal places
SQRTA(.., ..)	Square root of (a^2+b^2)
SQRTS(.., ..)	Square root of $(a^2–b^2)$

Variable types

The Control distinguishes the following variable types:

General variables

- **#l1 .. #l30 Channel-independent local variables** are effective within a main or a subprogram.
- **#c1 .. #c30 Channel-dependent, global variables** can be used for each slide (NC channel). Identical variable numbers on different slides are no problem. The variable content is provided globally by one channel; globally means that a variable described in a subprogram can be evaluated in the main program, and vice versa.
- **#g1 .. #g199 Channel-independent, global REAL variables** are provided once within the control. If the NC program changes a variable, it applies to all slides. The variables are retained even when the control is switched off, and can be evaluated again after power-up.
- **#g200 .. #g299 Channel-independent, global INTEGER variables** are provided once within the control. If the NC program changes a variable, it applies to all slides. The variables are retained even when the control is switched off, and can be evaluated again after power-up.
- **#x1 .. #x20 Channel-dependent, local text variables** are effective within a main or subprogram. They can only be read on the channel to which they were written.

If the variables are to be retained in the memory when the control is switched off, this feature must be activated by the machine tool builder (configuration parameter: “Channels/ChannelSettings/CH_NC1/CfgNcPgmParState/persistent=TRUE”).

If this feature is not activated, the variables values will always be "zero" after power-up.

Example:

```
. . .
N.. #l1=#l1+1
N.. G1 X#c1
N.. G1 X(SQRT(3*(SIN(30)))
N.. #g1=(ABS(#2+0.5))
. . .
N.. G1 Z#m(#l1)(Z)
N.. #x1=“Text“
N.. #g2=#g1+#l1*(27/9*3.1415)
. . .
```

Machine dimensions

- **#m1(n) .. #m9(n)** "n" is the designation of the axis (X, Z, Y) for which the machine dimension is to be read or written. The variable calculation uses the table "mach_dim.hmd".
  Simulation: During the startup of the control, the table "mach_dim.hmd" is read by the simulation. The simulation function now uses the table of the simulation.

Example: Machine dimensions

```
. . .
N.. G1 X(#m1(X)*2)
N.. G1 Z#m3(Z)
N.. #m4(Z)=350
. . .
```

Tool compensation

- **#dt(n):** "n" is the compensation direction (X, Z, Y, S) and "t" is the turret pocket number assigned to the tool. The variable calculation uses the table "toolturn.htt".
  Simulation: When the program is selected, the table "toolturn.htt" is read by the simulation. The simulation function now uses the table of the simulation.

Tool information can also be interrogated directly via the ID number. This may be necessary, for example, if no turret pocket has been assigned. For this purpose, program a comma and the ID number of the tool after the desired identification, e.g. `#L1 = #W1( ZL, "001")`.

Example: Tool compensation

```
. . .
N.. #d3(X)=0
N.. #d3(Z)=0.1
N.. #d3(S)=0.1
. . .
```

Event bits: Variable programming interrogates a bit of the event for 0 or 1. The meaning of the event is determined by the machine manufacturer.

- **#en(key):** "n" is the channel number and "key" is the event name. Used for reading external events set by the PLC.
- **#e0(key[n].xxx)** "n" is the channel number, "key" is the event name, and "xxx" is the name extension. Used for reading external events set by the PLC.

Example: Events

```
. . .
N.. #g1 = #e1( "NP_DG_axis_module_wait")
N.. PRINT( "NP_DG_axis_module_wait=",#g1)
N.. #g2 = #e1( "DG_DATA[1]")
N.. PRINT( "DG_DATA[1] =",#g2)
N.. #g3 = #e1( "SPI[1].DG_TEST[1]")
N.. PRINT( "SPI[1].DG_TEST[1] =",#g3)
. . .
N.. IF #e1( "NP_DG_axis_module_wait")==4
N.. THEN
N.. G0 X40 Z40
N.. ELSE
N.. G0 X60 Z60
N.. ENDIF
. . .
```

Reading tool data

Use the following syntax to read tool data. You can only access tools that are entered in the turret list.

If a sequence of exchange is defined, program the first tool of the sequence. The Control determines the data of the active tool.

> Tool information can also be interrogated directly via the ID number. This may be necessary, for example, if no turret pocket has been assigned. For this purpose, program a comma and the ID number of the tool after the desired identification, e.g. **`#L1 = #W1( ZL, "001")`**.

Identification codes for tool information	
#wn(ID)	Tool ID number (assign in text variable (#xn))
#wn(WT)	Tool type (3-digit number)
#wn(WTV)	1st position of tool type
#wn(WTH)	2nd position of tool type
#wn(WTL)	3rd position of tool type
#wn(NL)	Usable length (inside turning and drilling tools)
#wn(HR)	Main machining direction (see table at right)
#wn(NR)	Secondary machining direction of turning tools
#wn(AS)	Execution (see at right)
#wn(ZZ)	Number of teeth (milling tools)
#wn(RS)	Cutting radius
#wn(ZD)	Stud diameter
#wn(DF)	Cutter diameter
#wn(SD)	Shank diameter
#wn(SB)	Cutting width
#wn(SL)	Cutting length
#wn(AL)	Length of first cut
#wn(FB)	Cutter width
#wn(WL)	Tool orientation
#wn(ZL)	Tool setting dimension in Z
#wn(XL)	Tool setting dimension in X
#wn(YL)	Tool setting dimension in Y

Access to tool data of turret

Syntax: **#wn(select)**

- n = turret pocket number
- n = 0 for the current tool
- select = designates the information to be read

Main machining direction

#wn(HR) Primary machining directions:

- 0: Undefined
- 1: +Z
- 2: +X
- 3: –Z
- 4: –X
- 5: +/–Z
- 6: +/–X

Execution

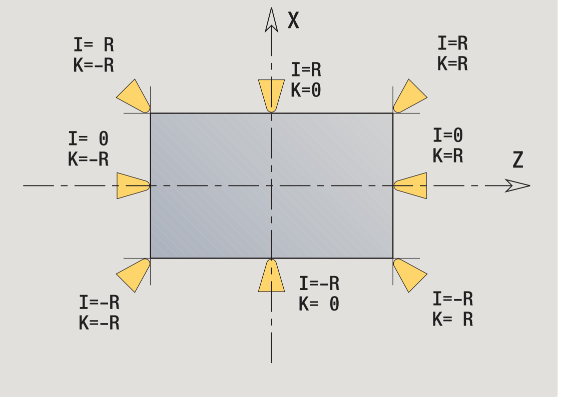
#wn(AS) Versions

- 1: Right-hand
- 2: Left-hand

Tool orientation

#wn(WL) Tool orientation (reference: machining direction of tool):

- 0: On the contour
- 1: To the right of the contour
- – 1: To the left of the contour



Identification codes for tool information	
#wn(I)	Position of tool tip center in X (see illustration)
#wn(J)	Position of tool tip center in Y
#wn(K)	Position of tool tip center in Z (see illustration)
#wn(ZE)	Distance between tool tip and slide zero point Z
#wn(XE)	Distance between tool tip and slide zero point X
#wn(YE)	Distance between tool tip and slide zero point Y
#wn(DN)	Diameter of drilling and milling tools
#wn(HW)	Principal angle in the normalized system (0° to 360°)
#wn(NW)	Secondary angle in the normalized system (0° to 360°)
#wn(EW)	Tool angle
#wn(SW)	Point angle
#wn(AW)	■ 0: No driven tool ■ 1: Driven tool
#wn(MD)	Direction of rotation: ■ 3: M3 ■ 4: M4
#wn(CW)	Tilting plane angle
#wn(BW)	Angular offset
#wn(WTL)	Orientation
#wn(AC)	Cutting-edge insert angle
#wn(ZS)	Maximum cutting depth
#wn(GH)	Thread pitch
#wn(NE)	Number of secondary cutting edges
#wn(NS)	Number of the secondary cutting edge
#wn(FP)	Tool type: 0 = normal tool, 1 = master tools, 2 = secondary cutting edge
#wn(Q)	Number of tool spindle
#wn(AS)	Execution left/right
#wn(DX)	Compensation in X
#wn(DY)	Compensation in Y
#wn(DZ)	Compensation in Z

Identification codes for tool information	
#wn(DS)	2nd compensation

Reading the current NC information

Use the following syntax to read NC information that was programmed with G functions.

Identification codes for NC information	
#n0(X)	Last programmed position X
#n0(Y)	Last programmed position Y
#n0(Z)	Last programmed position Z
#n0(C)	Last programmed position C
#n40(G)	Status of TRC (see table at right)
#n148(O)	Active wear compensation (see table at right)
#n18(G)	Active working plane (see table at right)
#n120(X)	Reference diameter X for calculating CY
#n52(G)	Oversize G52_Geo taken into account 0=no / 1=yes
#n57(X)	Oversize in X
#n57(Z)	Oversize in Z
#n58(P)	Equidistant oversize
#n150(X)	Cutting width shifted in X by G150/G151
#n150(Z)	Cutting width shifted in Z by G150/G151
#n95(G)	Programmed feed type (G93/G94/G95)
#n95(Q)	Spindle number of the last programmed feed rate
#n95(F)	Last programmed feed rate
#n97(G)	Programmed speed type (G96/G97)
#n97(Q)	Spindle number of the last programmed speed type
#n97(S)	Last programmed speed
#n47(P)	Current safety clearance
#n147(I)	Current safety clearance in working plane
#n147(K)	Current safety clearance in infeed direction

Access to current NC information	
Syntax:	**#nx(select)** ■ x = G function number ■ select = designates the information to be read

Status of TRC	
#n40(G)	TRC/MCRC status: ■ 40: G40 active ■ 41: G41 active ■ 42: G42 active

Active wear compensation	
#n148(O)	Active wear compensation (G148): ■ 0: DX, DZ ■ 1: DS, DZ ■ 2: DX, DS

Active working plane	
#n18(G)	Active working plane: ■ 17: XY plane (front or rear) ■ 18: XZ plane (turning) ■ 19: YZ plane (plan view / lateral surface)

Reading general NC information

Use the following syntax to read general NC information.

Identification codes for tool information	
#i1	Active operating mode (see table at right)
#i2	Active unit of measure (inches/metric)
#i3	■ Main spindle = 0 ■ Counterspindle with mirroring Z = 1 ■ Tool mirroring in Z = 2 ■ Tool + path mirroring in Z = 3
#i4	G16 active = 1 (currently not used)
#i5	Last programmed T number
#i6	Start block search active = 1
#i7	System is DataPilot = 1
#i8	Selected language
#i9	If Y axis is configured = 1
#i10	If B axis is configured = 1
#i11	If the tool pocket in X is mirrored to the machine system = 1
#i12	If U axis is programmable = 1
#i13	If V axis is programmable = 1
#i14	If W axis is programmable = 1
#i15	If U axis is configured = 1
#i16	If V axis is configured = 1
#i17	If W axis is configured = 1
#i18	Datum shift of the Z axis
#i19	Datum shift of the X axis
#i20	Last programmed path function (G0, G1, G2...)
#i21	Current quantity (workpiece counter)
#i99	Return code of subprograms

Active operating mode	
#i1	Active operating mode: ■ 2: Machine ■ 3: Simulation ■ 5: TSF menu

Active unit of measure	
#i2	Active unit of measure: ■ 0: Metric [mm] ■ 1: Inches [in]

Languages	
#i8	Available languages: ■ 0: ENGLISH ■ 1: GERMAN ■ 2: CZECH ■ 3: FRENCH ■ 4: ITALIAN ■ 5: SPANISH ■ 6: PORTUGUESE ■ 7: SWEDISH ■ 8: DANISH ■ 9: FINNISH ■ 10: DUTCH ■ 11: POLISH ■ 12: HUNGARIAN ■ 14: RUSSIAN ■ 15: CHINESE ■ 16: CHINESE_TRAD ■ 17: SLOVENIAN ■ 18: ESTONIAN ■ 19: KOREAN ■ 20: LATVIAN ■ 21: NORWEGIAN ■ 22: ROMANIAN ■ 23: SLOVAK ■ 24: TURKISH ■ 25: LITHUANIAN

Reading configuration data—PARA

The PARA function is used to read configuration data. To do this, use the parameter designations from the configuration parameters. You also use the designations from the configuration parameters to read user parameters.

When you read optional parameters, check whether the return value is valid. Depending on the data type of the parameter (REAL/STRING), the value "0" or the text "_EMPTY" is returned when reading an optional attribute that has not been set.

Access to configuration data	
Syntax:	**PARA(key, entity, attribute, index)**
	■ Key: Key word ■ Entity: Name of the configuration group ■ Attribute: Element name ■ Index: Array number if the attribute is from an array

Example: PARA function

`. . .`	
`N.. #l10=PARA("","CfgDisplayLanguage","ncLanguage")`	Reads the number of the currently selected language
`N.. #l1=PARA("","CfgGlobalTechPara","safetyDistWorkpOut")`	Reads the external safety clearance on the machined part (SAT)
`N.. #l1=PARA("Z1","CfgAxisProperties","threadSafetyDist")`	Reads the thread safety clearance for Z1
`N.. #l1=PARA("","CfgCoordSystem","coordSystem")`	Reads the machine orientation number
`. . .`	
`#x2=PARA("#x30","CfgCAxisProperties","relatedWpSpindle",0)`	Check whether the optional parameter is set
`IF #x2<>"_EMPTY"`	Evaluation:
` THEN`	
`  [ The parameter "relatedWpSpindle" was set ]`	
` ELSE`	
`  [ The parameter "relatedWpSpindle" was not set ]`	
`ENDIF`	

Determining the index of a parameter element—PARA

You activate the index search for an element by appending the name of the list element to the attribute, separated by a comma.

Example:

Determining the logical axis number of spindle S1

#c1 = PARA("", "CfgAxes", "axisList,S1", 0)

The function returns the index of the "S1" element in the "axisList" attribute of the "CfgAxes" entity. The index of element S1 equals the logical axis number in this example.

Without the attribute extension "S1", the function would read the element located at the list index number 0. Since the element is a string in this example, the result has to be assigned to a string variable.

#x1 = PARA("", "CfgAxes", "axisList", 0)

The function reads the string name of the element at list index number 0.

Access to configuration data	
Syntax:	**PARA("key","entity","attribute,element", index)**
	■ Key: Key word ■ Entity: Name of the configuration group ■ Attribute,name: Attribute name and element name ■ Index: 0 (not required)

Expanded variable syntax CONST – VAR

By defining the key words **CONST** or **VAR**, you can assign names to variables. The key words can be used in the main program and subprogram. To use the definitions in a subprogram, you need to declare the constant or variable before the **MACHINING** section code.

Rules for defining constants and variables:

The names of constants and variables must be preceded by an underscore. They can comprise lower case letters, numbers and the underscore character. The maximum length must not exceed 20 characters.

Variable names with VAR

By assigning variable names, you make it easier to read an NC program. To do this, you must insert the program section VAR. In this program section, you assign the variable designations to the variables.

Example: Free-text variables

```
%abc.nc
VAR
#_rohdm=#l1 [#_rohdm is a synonym of #l1]
BLANK
N..
FINISHED PART
N..
MACHINING
N..
. . .
```

Example: Subprogram

```
%SP1.ncS
VAR
#_wo = #c1    [tool orientation]
MACHINING
N.. #_wo = #w0(WTL)
N.. G0 X(#_posx*2)
N.. G0 X#_start_x
. . .
```

Definition of constants—CONST

Possibilities of defining constants:

- Direct assignment of values
- Internal interpreter information as constants
- Assignment of names to the transfer variables of subprograms

Use the following internal information to define constants in the CONST section.

Internal information for defining constants	
__n0_x	768 Last programmed position X
__n0_y	769 Last programmed position Y
__n0_z	770 Last programmed position Z
__n0_c	771 Last programmed position C
__n40_g	774 Status of TRC
__n148_o	776 Active wear compensation
__n18_g	778 Active working plane
__n120_x	787 Reference diameter X for calculating CY
__n52_g	790 Oversize G52_Geo taken into account 0=no / 1=yes
__n57_x	791 Oversize in X
__n57_z	792 Oversize in Z
__n58_p	793 Equidistant oversize
__n150_x	794 Cutting width shifted in X by G150/G151
__n150_z	795 Cutting width shifted in Z by G150/G151
__n95_g	799 Programmed feed type _G93/G94/G95)
__n95_q	796 Spindle number of the programmed feed rate
__n95_f	800 Last programmed feed rate
__n97_g	Programmed speed type _G96/G97)
__n97_q	797 Spindle number of the programmed speed type
__n97_s	Last programmed speed
__la-__z	Subprogram transfer values

The constant "_pi" is predefined to the value 3.1415926535989 and can be used directly in every NC program.

Example: Main program

```
%abc.nc
CONST
_square_root2 = 1.414213 [direct value assignment]
_square_root_2 = SQRT(2) [direct value assignment]
_posx = __n0_x     [internal information]
VAR
. . .
BLANK
N..
FINISHED PART
N..
MACHINING
N..
. . .
```

Example: Subprogram

```
%SP1.ncS
CONST
_start_x=__la [subprogram transfer value]
_posx = __n0_x      [internal constant]
VAR
#_wo = #c1    [tool orientation]
MACHINING
N.. #_wo = #w0(WTL)
N.. G0 X(#_posx*2)
N.. G0 X#_start_x
. . .
```

4.32 Conditional block run

Program branching IF..THEN..ELSE..ENDIF

A conditional branch consists of the elements:

- IF, followed by a condition. The condition includes a variable or mathematical expression on either side of the relational operator.
- THEN. If the condition is fulfilled, the THEN branch is executed.
- ELSE. If the condition is not fulfilled, the ELSE branch is executed.
- ENDIF concludes the conditional program branch.

Interrogate bitset: You can also use the BITSET function as condition. The function returns 1 if the numerical value contains the requested bit. The function returns 0 if the numerical value does not contain the requested bit.

Syntax: **BITSET (x,y)**

- x: Bit number (0 to 15)
- y: Numerical value (0 to 65535)

The relationship between bit number and numerical value is shown in the table at right. You can also use variables for x, y.

Programming:

- Select "Extras > DINplus word...". The Control opens the "Insert DIN PLUS word" selection list.
- Select IF.
- Enter the condition.
- Insert NC blocks of the THEN branch.
- If required: Insert NC blocks of the ELSE branch.

> - NC blocks with IF, THEN, ELSE, ENDIF can have no further commands.
> - You can combine up to two conditions.

Relational operators	
<	Less than
<=	Less than or equal to
<>	Not equal to
>	Greater than
>=	Greater than or equal to
==	Equal to
Combining conditions:	
AND	Logical AND operation
OR	Logical OR operation

Bit	which means numerical value	Bit	which means numerical value
0	1	8	256
1	2	9	512
2	4	10	1024
3	8	11	2048
4	16	12	4096
5	32	13	8192
6	64	14	16384
7	128	15	32768

Example: IF..THEN..ELSE..ENDIF

```
N.. IF (#l1==1) AND (#g250>50)
N.. THEN
N..     G0 X100 Z100
N.. ELSE
N..     G0 X0 Z0
N.. ENDIF
. . .
N.. IF 1==BITSET(0,#l1)
N.. THEN
N..   PRINT("bit 0: OK")
. . .
```

Requesting variables and constants

With the DEF, NDEF, and DVDEF elements you can inquire whether a valid value was assigned to a variable or a constant. For example, an undefined variable can return the value 0, just like a variable that has been assigned the value 0. You can prevent undesired program jumps by examining the variables.

Programming:

- Select "Extras > DINplus word...". The Control opens the "Insert DIN PLUS word" selection list.
- Select the IF command
- Enter the required inquiry element (DEF, NDEF or DVDEF)
- Enter the name of a variable or a constant

Enter the variable name without the character "#", e.g. **IF NDEF(__la)**.

Inquiry elements of variables and constants:

- DEF: A value is assigned to a variable or constant
- NDEF: No value is assigned to a variable or constant
- DVDEF: Inquiry of an internal constant

Example: Requesting variable in subprogram

```
N.. IF DEF(__la)
N.. THEN
N.. PRINT(„Value:“,#__la)
N.. ELSE
N.. PRINT(„#__la is not defined“)
N.. ENDIF
. . .
```

Example: Requesting variable in subprogram

```
N.. IF NDEF(__lb)
N.. THEN
N.. PRINT(„#__lb is not defined“)
N.. ELSE
N.. PRINT(„Value:“,#__lb)
N.. ENDIF
. . .
```

Example: Requesting constants

```
N.. IF DVDEF(__n97_s)
N.. THEN
N.. PRINT(„__n97_s is defined“,#__n97_s)
N.. ELSE
N.. PRINT(„#__n97_s is not defined“)
N.. ENDIF
. . .
```

WHILE..ENDWHILE program repeat

A program repeat consists of the elements:

- WHILE, followed by a condition. The condition includes a variable or mathematical expression on either side of the relational operator.
- ENDWHILE concludes the conditional program repeat.

NC blocks programmed between WHILE and ENDWHILE are executed repeatedly for as long as the condition is fulfilled. If the condition is not fulfilled, the Control continues execution of the program with the block programmed after ENDWHILE.

Interrogate bitset: You can also use the BITSET function as condition. The function returns 1 if the numerical value contains the requested bit. The function returns 0 if the numerical value does not contain the requested bit.

Syntax: **BITSET (x,y)**

- x: Bit number (0 to 15)
- y: Numerical value (0 to 65535)

The relationship between bit number and numerical value is shown in the table at right. You can also use variables for x, y.

Programming:

- Select "Extras > DINplus word...". The Control opens the "Insert DIN PLUS word" selection list.
- Select WHILE.
- Enter the condition.
- Insert NC blocks between WHILE and ENDWHILE.

> - You can combine up to two conditions.
> - If the condition you program in the WHILE command is always true, the program remains in an endless loop. This is one of the most frequent causes of error when working with program repeats.

Relational operators	
<	Less than
<=	Less than or equal to
<>	Not equal to
>	Greater than
>=	Greater than or equal to
==	Equal to
Combining conditions:	
AND	Logical AND operation
OR	Logical OR operation

Bit	which means numerical value	Bit	which means numerical value
0	1	8	256
1	2	9	512
2	4	10	1024
3	8	11	2048
4	16	12	4096
5	32	13	8192
6	64	14	16384
7	128	15	32768

Example: WHILE..ENDWHILE

```
. . .
N.. WHILE (#l4<10) AND (#l5>=0)
N..     G0 Xi10
. . .
N.. ENDWHILE
. . .
```

SWITCH..CASE—program branching

The switch statement consists of the elements:

- SWITCH, followed by a variable. The content of the variable is interrogated in the following CASE statement.
- CASE x: The CASE branch is run with the variable value x. CASE can be programmed repeated times.
- DEFAULT: This branch is executed if no CASE statement matched the variable value. DEFAULT can be omitted.
- BREAK: Concludes the CASE branch or DEFAULT branch.

Programming:

- Select "Extras > DINplus word...". The Control opens the "Insert DIN PLUS word" selection list.
- Select SWITCH.
- Enter the switch variable.
- For each CASE branch:
  - Select CASE (in "Extras > DINplus word...").)
  - Enter the SWITCH condition (value of the variable) and insert the NC blocks to be executed.
- For the DEFAULT branch: Insert the NC blocks to be executed.

Example: SWITCH..CASE

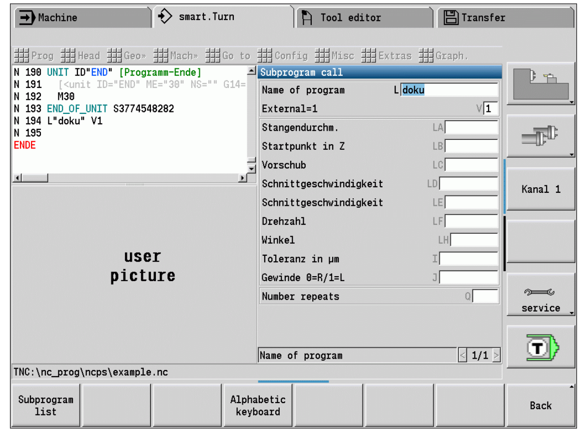
. . .	
N.. SWITCH #g201	
N.. CASE 1 [executed if #g201=1]	Executed if #g201=1
N.. G0 Xi10	
. . .	
N.. BREAK	
N.. CASE 2 [executed if #g201=2]	Executed if #g201=2
N.. G0 Xi20	
. . .	
N.. BREAK	
N.. DEFAULT	No CASE statement matched the variable value
N.. G0 Xi30	
. . .	
N.. BREAK	
N.. ENDSWITCH	
. . .	

4.33 Subroutines

Subprogram call: L"xx" V1

The subprogram contains the following elements:

- L: Identifying letter for subprogram call
- "xx": Name of the subprogram—file name for external subprograms (max. 16 letters or numbers)
- V1: Identification code for **external** subprograms—omitted for local subprograms



Note on using subprograms:

- External subprograms are defined in a separate file. They can be called from any main program or other subprogram.
- Local subprograms are in the main program file. They can be called only from the main program.
- Subprograms can be nested up to 6 times. Nesting means that another subprogram is called from within a subprogram.
- Recursion should be avoided.
- You can add up to 29 transfer values to a subprogram.
  - Designations: LA to LF, LH, I, J, K, O, P, R, S, U, W, X, Y, Z, BS, BE, WS, AC, WC, RC, IC, KC and JC
  - The identification code within the subprogram is: #__.., followed by the parameter designation in lowercase letters (for example: #__la).
  - Use the transfer values when programming with variables within the subprogram.
  - String variables: ID and AT
- The variables #I1 – #I30 are available in every subprogram as local variables.
- To transfer a variable to the main program, program the variable after the fixed word RETURN. In the main program, the information is available in #i99.
- If a subprogram is to be executed repeatedly, define in the "number of repeats" Q parameter the number of times the subprogram is to be repeated.
- A subprogram ends with RETURN.

The parameter LN is reserved for the transfer of block numbers. This parameter may receive a new value when the NC program is renumbered.

Dialog texts in subprogram call

You can define up to 30 parameter descriptions that precede/follow the input fields in an external subprogram. The units of measure are defined using code numbers. Depending on the setting "metric" or "inches," the Control shows the designations (of the units of measure). When calling external subprograms that contain a parameter list, then parameters not contained in this list are omitted from the call dialog.

The parameter descriptions can be positioned within the subprogram as desired. The control searches for subprograms in the sequence: current project, standard directory and then machine manufacturer directory.

Parameter descriptions (see table at right):

[//] – Beginning

[pn=n; s=parameter text (up to 25 characters)]

[//] – End

pn: Parameter designations (la, lb, ...)

n: Code number for units of measure

- 0: Non-dimensional
- 1: mm or inches
- 2: mm/rev or in./rev
- 3: mm/min or in./min
- 4: m/min or ft/min
- 5: Rev/min
- 6: Degrees (°)
- 7: µm or µinch

Example:

```
. . .
[//]
[la=1; s=bar diameter]
[lb=1; s=starting point in Z]
[lc=1; s=chamfer/rounding arc (-/+)]
. . .
[//]
. . .
```

Help graphics for subprogram calls

With help graphics you illustrate the calling parameters of subprograms. The Control places the help graphics to the left next to the dialog box of the subprogram call.

If you append an underscore "_" and the input field name in capital letters (beginning with L) to the name of the file name, a separate graphic will be displayed for that input field. If no separate help graphic exists for an input field, the graphic for the subprogram will be displayed (if available). In the standard setting, the help window is displayed only if there is a graphic for the subprogram. Even if you want to use individual graphics for the address letters, you should define a graphic for the subprogram.

Graphic format:

- BMP, PNG, JPG images
- Size 440x320 pixels

You integrate help graphics for subprogram calls as follows:

- ▶ The subprogram name, entry field name and the appropriate extension (BMP, PNG, JPG) must be used as file name for the help graphic.
- ▶ Transfer the help graphic to the directory: \nc_prog\Pictures

4.34 M commands

M commands for program-run control

The effect of machine commands depends on the configuration of your machine. On your lathe, other M commands may apply for the listed functions. Refer to your machine manual.

Overview: M commands for program-run control	
M00	**Program stop** The program run stops. **Cycle start** resumes the program run.
M01	**Optional stop** If the **Continuous run** soft key is not active in Automatic mode, the program run stops with M01. **Cycle start** resumes the program run. If **Continuous run** is active, the program continues without stopping.
M18	**Counting pulse**
M30	**End of program** M30 means "end of program" (you do not need to program M30). If you press **Cycle Start** after M30, program execution is repeated from the start of the program.
M417	Activate protection zone monitoring
M418	Deactivate protection zone monitoring
M99 NS..	Program end with restart M99 means end program and start again. Control restarts program execution from: ■ The start of program if no NS is entered ■ The block number NS if a NS is entered

Modal functions (feed rate, spindle speed, tool number, etc.) which are effective at the end of program remain in effect when the program is restarted. You should therefore reprogram the modal functions at the start of program or at the startup block (if M99 is used).

Machine commands

The effect of machine commands depends on the configuration of your machine. The following table lists the M commands used on most machines.

M commands as machine commands	
M03	Main spindle on (cw)
M04	Main spindle on (ccw)
M05	Main spindle stop
M12	Lock main spindle brake
M13	Release main spindle brake
M14	C axis on
M15	C axis off
M19..	Spindle stop at position C
M40	Shift gear to range 0 (neutral)
M41	Shift gear to range 1
M42	Shift gear to range 2
M43	Shift gear to range 3
M44	Shift gear to range 4
Mx03	Spindle x on (cw)
Mx04	Spindle x on (ccw)
Mx05	Spindle x stop

For more information on the M commands, refer to your machine manual.

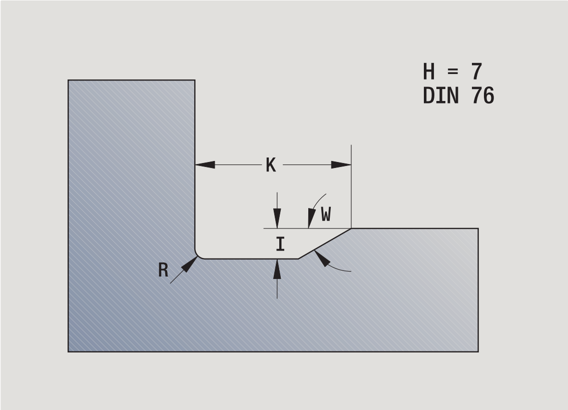
4.35 G functions from previous controls

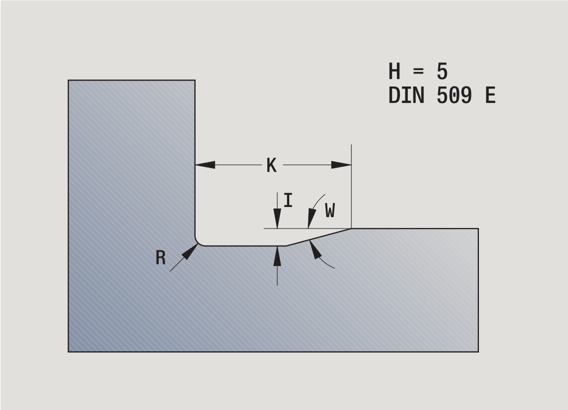
The commands described in the following are supported to enable you to use NC programs from previous controls. HEIDENHAIN recommends against using these commands in new NC programs.

Contour definitions in the machining section

Undercut contour G25

G25 generates an undercut form element (DIN 509 E, DIN 509 F, DIN 76) that can be integrated in the contour description of roughing or finishing cycles. The help graphic illustrates the undercut parameters.





Parameters

- H Undercut type (default: 0)
  - H=0, 5: DIN 509 E
  - H=6: DIN 509 F
  - H=7: DIN 76
- I Undercut depth (default: value from standard table)
- K Undercut width (default: value from standard table)
- R Undercut radius (default: value from standard table)
- P Face depth (default: value from standard table)
- W Undercut angle (default: value from standard table)
- A Face angle (default: value from standard table)
- FP Thread pitch—no value: Pitch calculated from thread diameter
- U Grinding oversize (default: 0)
- E Reduced feed for machining the undercut (default: active feed rate)

If the parameters are not defined, the Control determines the following values from the diameter or the thread pitch in the standard table:

- DIN 509 E: I, K, W, R
- DIN 509 F: I, K, W, R, P, A
- DIN 76: I, K, W, R (determined from the thread pitch)

- All parameters that you enter will be accounted for—even if the standard table prescribes other values.
- If you are programming an internal thread, it is advisable to preset the **thread pitch FP** since the diameter of the longitudinal element is not the thread diameter. If you have Control calculate the thread pitch automatically, slight deviations may occur.

Example: G25

```
%25.nc
[G25]
N1 T1 G95 F0.4 G96 S150 M3
N2 G0 X62 Z2
N3 G819 P4 H0 I0.3 K0.1
N4 G0 X13 Z0
N5 G1 X16 Z-1.5
N6 G1 Z-30
N7 G25 H7 I1.15 K5.2 R0.8 W30 FP1.5
N8 G1 X20
N9 G1 X40 Z-35
N10 G1 Z-55 B4
N11 G1 X55 B-2
N12 G1 Z-70
N13 G1 X60
N14 G80
END
```

Simple turning cycles

Simple longitudinal roughing G81

G81 roughs the contour area defined by the current tool position and X, Z. If you wish to machine an oblique cut, you can define the angle with I and K.

Parameters

X Starting point of contour in X (diameter value)
Z Contour end point
I Maximum infeed in X
K Offset in Z direction (default: 0)
Q G function for infeed (default: 0)

- 0: Infeed with G0 (rapid traverse)
- 1: Infeed with G1 (feed rate)

V Type of retraction (default: 0)

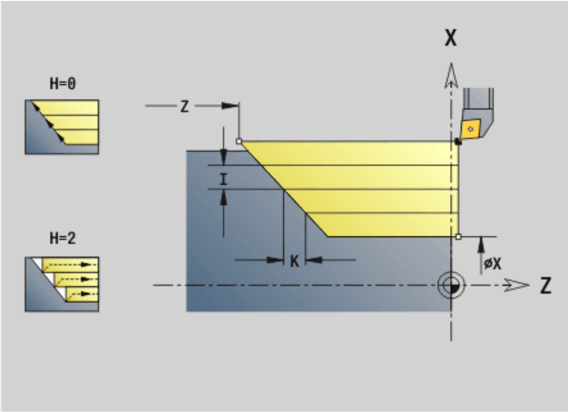
- 0: Return to cycle starting point in Z and last retraction diameter in X
- 1: Return to cycle starting point

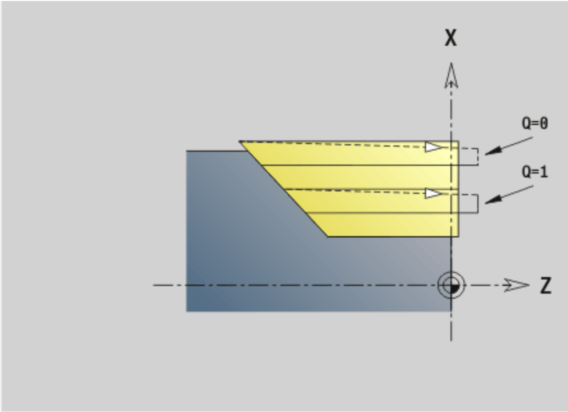
H Type of departure (default: 0)

- 0: With each cut (machine contour outline after each pass)
- 2: No smoothing (retracts at 45°; no contour smoothing)

The Control uses the position of the target point to distinguish between external and internal machining. The number of cutting passes is calculated so that an abrasive cut is avoided and the calculated infeed distance is <= I.

- **Programming X, Z:** Absolute, incremental or modal
- The **tool radius compensation** is not active.
- **Safety clearance** after each pass: 1 mm
- A **G57 oversize**
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58 oversize** is not taken into account.





Example: G81

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G81 X100 Z-70 I4 K4 Q0
N4 G0 X100 Z2
N5 G81 X80 Z-60 I-4 K2 Q1
N6 G0 X80 Z2
N7 G81 X50 Z-45 I4 Q1
. . .
```

Simple face roughing G82

G82 roughs the contour area defined by the current tool position and X, Z. If you wish to machine an oblique cut, you can define the angle with I and K.

Parameters

X Contour end point in X (diameter value)
Z Contour starting point
I Offset in X direction (default: 0)
K Maximum infeed in Z
Q G function for infeed (default: 0)

- 0: Infeed with G0 (rapid traverse)
- 1: Infeed with G1 (feed rate)

V Type of retraction (default: 0)

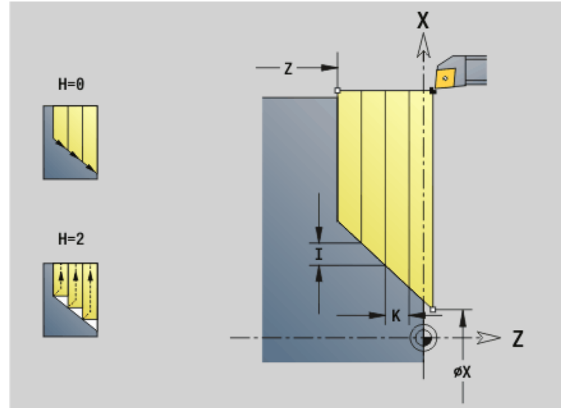
- 0: Return to cycle starting point in X and last retraction position in Z
- 1: Return to cycle starting point

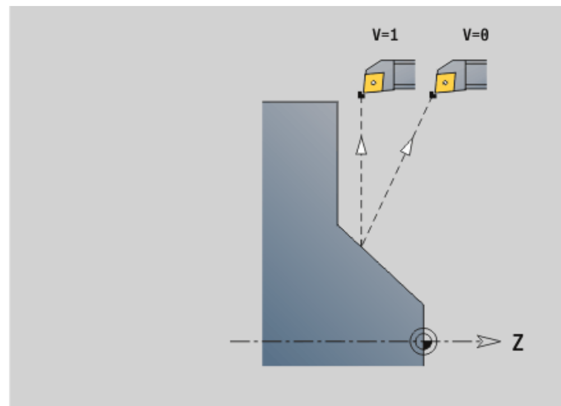
H Type of departure (default: 0)

- 0: With each cut (machine contour outline after each pass)
- 2: No smoothing (retracts at 45°; no contour smoothing)

The Control uses the position of the target point to distinguish between external and internal machining. The number of cutting passes is calculated so that an abrasive cut is avoided and the calculated infeed distance is <= K.

- **Programming X, Z:** Absolute, incremental or modal
- The **tool radius compensation** is not active.
- **Safety clearance** after each pass: 1 mm
- A **G57 oversize**
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58 oversize** is not taken into account.





Example: G82

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G82 X20 Z-15 I4 K4 Q0
N4 G0 X120 Z-15
N5 G82 X50 Z-26 I2 K-4 Q1
N6 G0 X120 Z-26
N7 G82 X80 Z-45 K4 Q1
. . .
```

Simple contour repeat cycle G83

G83 carries out the functions programmed in the following blocks (simple traverses or cycles without a contour definition) more than once. G80 ends the machining cycle.

Parameters

X	Contour target point (diameter) – (default: Load the last X coordinate)
Z	Contour target point – (default: Load the last Z coordinate)
I	Maximum infeed in X direction (radius)—(default: 0)
K	Maximum infeed in Z direction (default: 0)

If the number of infeeds differs for the X and Z axes, the tool first advances in both axes with the programmed values. The infeed is set to zero if the target value for one direction is reached.

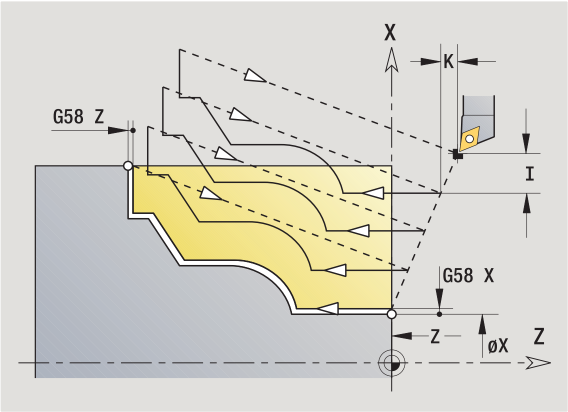
Programming:

- G83 is alone in the block
- G83 must not be nested, not even by calling subprograms

- The **tool radius compensation** is not active. You can program the TRC separately with G40 to G42.
- **Safety clearance** after each pass: 1 mm
- A **G57 oversize**
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58 oversize**
  - Is taken into account if you work with TRC
  - Remains effective after cycle end

Danger of collision!

After each pass, the tool returns on a diagonal path before it advances for the next pass. If required, program an additional rapid traverse path to avoid a collision.

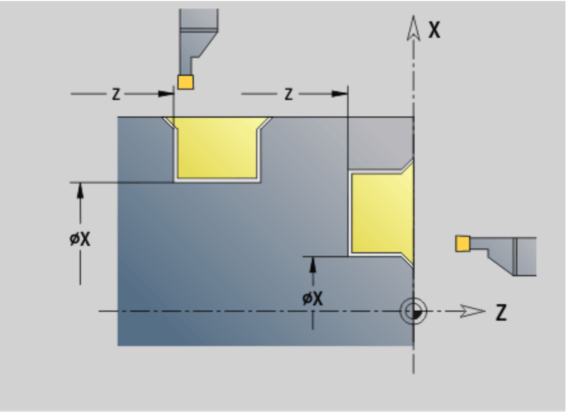


Example: G83

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G83 X80 Z0 I4 K0.3
N4 G0 X80 Z0
N5 G1 Z-15 B-1
N6 G1 X102 B2
N7 G1 Z-22
N8 G1 X90 Zi-12 B1
N9 G1 Zi-6
N10 G1 X100 A80 B-1
N11 G1 Z-47
N12 G1 X110
N13 G0 Z2
N14 G80
```

Recessing G86

G86 machines simple radial and axial recesses with chamfers. From the tool position, the Control determines whether a radial or axial recess, or an inside or outside recess is to be machined.



Parameters

X Base corner point (diameter)

Z Base corner point

I Radial recess: Oversize

- I>0: Oversize (roughing and finishing)
- I=0: No finishing

Axial recess: Recess width

- I>0: Recess width
- No input: Recess width = tool width

K Radial recess: Recess width

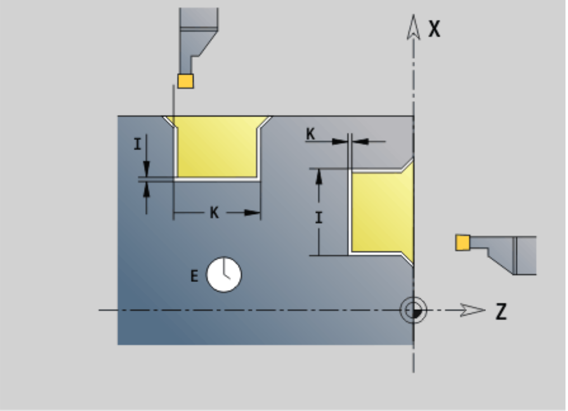
- K>0: Recess width
- No input: Recess width = tool width

Axial recess: Oversize

- K>0: Oversize (roughing and finishing)
- K=0: No finishing

E Dwell time (for chip breaking)—(default: length of time for one revolution)

- With finishing oversize: Only for finishing
- Without finishing oversize: For every recess



"Oversize" programmed: First roughing, then finishing

G86 machines chamfers at the sides of the recess. If you do not wish to cut the chamfers, you must position the tool at a sufficient distance from the workpiece. Calculate the starting position XS (diameter) as follows:

XS = XK + 2 * (1.3 – b)

XK: Contour diameter

b: Chamfer width

- The **tool radius compensation:** is active.
- **Oversizes** are not taken into account.

Example: G86

```
. . .
N1 T30 G95 F0.15 G96 S200 M3
N2 G0 X62 Z2
N3 G86 X54 Z-30 I0.2 K7 E2 [radial]
N4 G14 Q0
N5 T38 G95 F0.15 G96 S200 M3
N6 G0 X120 Z1
N7 G86 X102 Z-4 I7 K0.2 E1 [axial]
. . .
```

Radius cycle G87

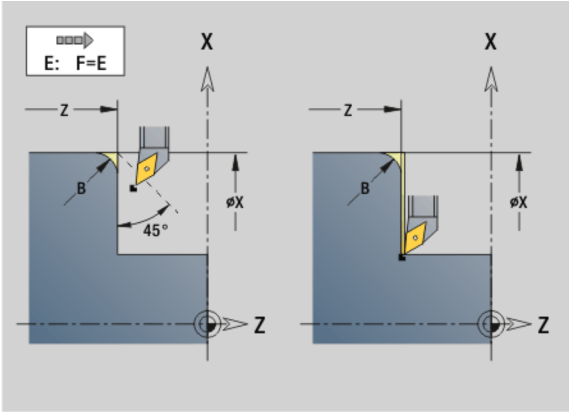
G87 machines transition radii at orthogonal, paraxial inside and outside corners. The direction is taken from the position/machining direction of the tool.

Parameters

- X Corner point (diameter)
- Z Corner point
- B Radius
- E Reduced feed rate (default: active feed)

A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed.

- The **tool radius compensation:** is active.
- **Oversizes** are not taken into account.



Example: G87

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G1 Z0
N4 G87 X84 Z0 B2 [radius]
```

Chamfer cycle G88

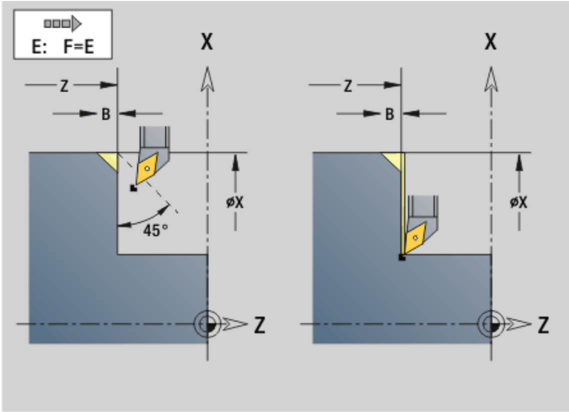
G88 machines chamfers at orthogonal, paraxial outside corners. The direction is taken from the position/machining direction of the tool.

Parameters

- X Corner point (diameter)
- Z Corner point
- B Chamfer width
- E Reduced feed rate (default: active feed)

A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed.

- The **tool radius compensation:** is active.
- **Oversizes** are not taken into account.



Example: G88

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G1 Z0
N4 G88 X84 Z0 B2 [chamfer]
```

Thread cycles (4110)

Simple longitudinal single-start thread G350

G350 cuts a longitudinal thread (internal or external). The thread starts at the current tool position and ends at the end point Z.

Parameters

Z	Corner point of thread
F	Thread pitch
U	Thread depth ■ U>0: Internal thread ■ U<0: External thread ■ U= +999 or –999: Thread depth is calculated
I	Maximum infeed (no input: I is calculated from the thread pitch and the thread depth)

Internal or external threads: See algebraic sign of "U."

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depending on the current cutting depth without exceeding the starting and end points of the thread.
- **Z direction:** Maximal 1 thread groove, without exceeding the starting and end points of the thread.

- **Cycle stop** becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle run.
- Handwheel superimpositioning can be activated with a switch located on the machine operating panel if your machine is equipped accordingly.
- **Feedforward control** is switched off.

Simple longitudinal multi-start thread G351

G351 machines a single or multi-start longitudinal thread (internal or external thread) with variable pitch. The thread starts at the current tool position and ends at the end point Z.

Parameters

- Z Corner point of thread
- F Thread pitch
- U Thread depth
  - U>0: Internal thread
  - U<0: External thread
  - U= +999 or –999: Thread depth is calculated
- I Maximum infeed (no input: I is calculated from the thread pitch and the thread depth)
- A Approach (infeed) angle (default 30°, range: –60° < A < 60°)
  - A>0: Infeed on right thread flank
  - A<0: Infeed on left thread flank
- D Threads per unit (default: 1)
- J Remaining cutting depth (default: 1/100 mm)
- E Variable pitch (default: 0)
  - E>0: Increases the pitch per revolution by E
  - E<=: Decreases the pitch per revolution by E

Internal or external threads: See algebraic sign of "U."

Number of cutting passes: "I" is used for the first pass. The cutting depth is reduced with each further pass until "the remaining cutting depth J" is reached.

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depending on the current cutting depth without exceeding the starting and end points of the thread.
- **Z direction:** Maximal 1 thread groove, without exceeding the starting and end points of the thread.

- **Cycle stop** becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle run.
- Handwheel superimpositioning can be activated with a switch located on the machine operating panel if your machine is equipped accordingly.
- **Feedforward control** is switched off.

4.36 DINplus program example

Example of a subprogram with contour repetitions

Contour repetitions, including saving of the contour

HEADER	
#SLIDE $1	
TURRET 1	
T2 ID "121-55-040.1"	
T3 ID "111-55.080.1"	
T4 ID "161-400.2"	
T8 ID "342-18.0-70"	
T12 ID "112-12-050.1"	
BLANK	
N1 G20 X100 Z120 K1	
Finished part	
N2 G0 X19.2 Z-10	
N3 G1 Z-8.5 BR0.35	
N4 G1 X38 BR3	
N5 G1 Z-3.05 BR0.2	
N6 G1 X42 BR0.5	
N7 G1 Z0 BR0.2	
N8 G1 X66 BR0.5	
N9 G1 Z-10 BR0.5	
N10 G1 X19.2 BR0.5	
MACHINING	
N11 G26 S2500	
N12 G14 Q0	
N13 G702 Q0 H1	Save contour
N14 L"1" V0 Q2	Qx = number of repetitions
N15 M30	
SUBPROGRAM "1"	
N16 M108	
N17 G702 Q1 H1	Load saved contour

N18 G14 Q0	
N19 T8	
N20 G97 S2000 M3	
N21 G95 F0.2	
N22 G0 X0 Z4	
N23 G147 K1	
N24 G74 Z-15 P72 I8 B20 J36 E0.1 K0	
N25 G14 Q0	
N26 T3	
N27 G96 S300 G95 F0.35 M4	
N28 G0 X72 Z2	
N29 G820 NS8 NE8 P2 K0.2 W270 V3	
N30 G14 Q0	
N31 T12	
N32 G96 S250 G95 F0.22	
N33 G810 NS7 NE3 P2 I0.2 K0.1 Z-12 H0 W180 Q0	
N34 G14 Q2	
N35 T2	
N36 G96 S300 G95 F0.08	
N37 G0 X69 Z2	
N38 G47 P1	
N39 G890 NS8 V3 H3 Z-40 D3	
N40 G47 P1	
N41 G890 NS9 V1 H0 Z-40 D1 I74 K0	
N42 G14 Q0	
N43 T12	
N44 G0 X44 Z2	
N45 G890 NS7 NE3	
N46 G14 Q2	
N47 T4	Insert parting tool
N48 G96 S160 G95 F0.18 M4	
N49 G0 X72 Z-14	
N50 G150	Shift reference point to the right of the cutting edge
N51 G1 X60	
N52 G1 X72	
N53 G0 Z-9	
N54 G1 X66 G95 F0.18	
N55 G42	Activate TRC

N56 G1 Z-10 B0.5	
N57 G1 X17	
N58 G0 X72	
N59 G0 X80 Z-10 G40	Deactivate TRC
N60 G14 Q0	
N61 G56 Z-14.4	Incremental zero point shift
RETURN	
END	

4.37 Connection between geometry and machining commands

Turning

Function	Geometry	Machining
Individual elements	■ G0..G3 ■ G12/G13	■ G810 Longitudinal roughing cycle ■ G820 Face roughing cycle ■ G830 Contour-parallel roughing cycle ■ G835 Contour-parallel with neutral tool ■ G860 Universal recessing cycle ■ G869 Recess turning cycle ■ G890 Finishing cycle
Recess	■ G22 (standard)	■ G860 Universal recessing cycle ■ G870 Simple recessing cycle ■ G869 Recess turning cycle
Recess	■ G23	■ G860 Universal recessing cycle ■ G869 Recess turning cycle
Thread with undercut	■ G24	■ G810 Longitudinal roughing cycle ■ G820 Face roughing cycle ■ G830 Contour-parallel roughing cycle ■ G890 Finishing cycle ■ G31 Thread cycle
Undercut	■ G25	■ G810 Longitudinal roughing cycle ■ G890 Finishing cycle
Thread	■ G34 (standard) ■ G37 (general)	■ G31 Thread cycle
Hole	■ G49 (turning center)	■ G71 Simple drilling cycle ■ G72 Counterboring, countersinking, etc. ■ G73 Tapping cycle ■ G74 Deep hole drilling cycle

C-axis machining—front/rear face

Function	Geometry	Machining
Individual elements	■ G100 to G103	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Figures	■ G301 Linear slot ■ G302/G303 Circular slot ■ G304 Full circle ■ G305 Rectangle ■ G307 Eccentric polygon	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Hole	■ G300	■ G71 Simple drilling cycle ■ G72 Counterboring, countersinking, etc. ■ G73 Tapping cycle ■ G74 Deep hole drilling cycle

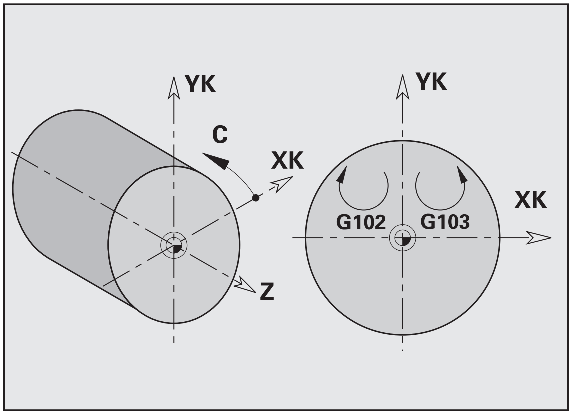
C-axis machining—lateral surface

Function	Geometry	Machining
Individual elements	■ G110 to G113	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Figures	■ G311 Linear slot ■ G312/G313 Circular slot ■ G314 Full circle ■ G315 Rectangle ■ G317 Eccentric polygon	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Hole	■ G310	■ G71 Simple drilling cycle ■ G72 Counterboring, countersinking, etc. ■ G73 Tapping cycle ■ G74 Deep hole drilling cycle

4.38 Full-surface machining

Fundamentals of full-surface machining

In "full-surface machining," the front and rear ends can be machined in **one** NC program. The control supports full-surface machining for all common machine designs. The features include angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation. This ensures efficient full-surface machining and simple programming.

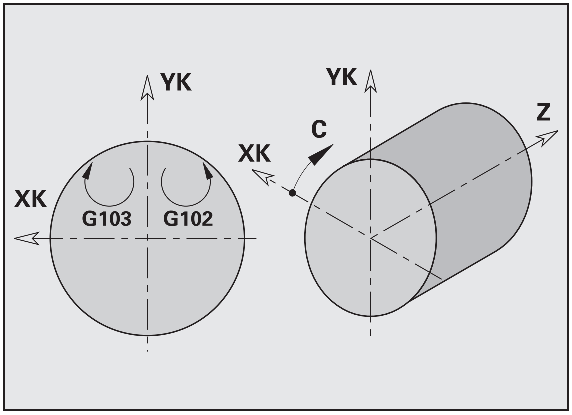


You describe the turning contour, the contours for the C axis as well as full-surface machining functions in one NC program. Expert programs are available for configuring the lathe.

You can also enjoy the benefits of full-surface machining on lathes with only one spindle.

Rear-face contours with C axis: The XK axis and therefore also the C axis are oriented with respect to the workpiece, not to the spindle. Therefore, for the rear face:

- Orientation of the XK axis: To the left (front face: to the right)
- Orientation of the C axis: Clockwise
- Direction of rotation for circular arcs G102: Counterclockwise.
- Direction of rotation for circular arcs G103: Clockwise.



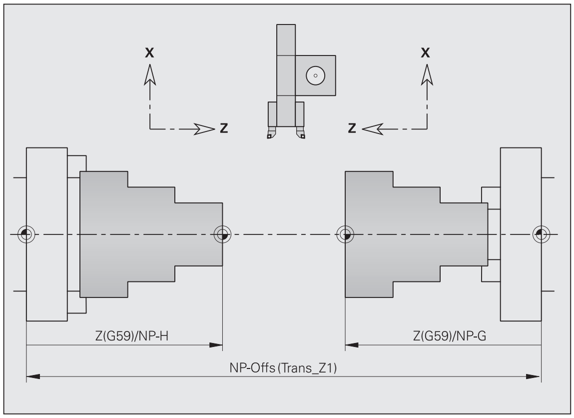
Turning: The CNC PILOT supports full-surface machining with conversion and mirroring functions. This makes it possible to keep the usual directions of movement for rear-side machining as well.

- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.

The machine manufacturer can provide your lathe with suitable **expert programs** for workpiece transfer.

Reference points and coordinate system: The position of the machine and workpiece zero points as well as the coordinate systems for the spindle and opposing spindle are illustrated in the figure at bottom. With this design of lathe it is recommended to mirror only the Z axis. Then, for either spindle, moving in positive direction will stand for motion away from the workpiece.

Usually the expert program contains the mirroring of the Z axis and the zero-point shift by the dimension "NP-Offs."



Programming of full-surface machining

When programming a contour on the rear face, be sure to consider the orientation of the XK axis (or X axis) and rotational direction of arcs.

Insofar as you use drilling and milling cycles, there are no special aspects to rear-face machining, since these cycles refer to predefined contours.

For rear-face machining with the basic commands G100 to G103 the same conditions apply as for rear-face contours.

Turning operations: The expert programs for reclamping contain converting and mirroring functions. The following principle applies for rear-face machining (2nd setup):

- + direction: Goes away from the workpiece
- – direction: Goes toward the workpiece
- G2/G12: Circular arc clockwise
- G3/G13: Circular arc counterclockwise

Working without expert programs
If you do not use the expert programs or the converting and mirroring functions, the following principle applies:

- **+ direction:** Goes away from the spindle
- **– direction:** Goes toward the main spindle
- **G2/G12:** Circular arc clockwise
- **G3/G13:** Circular arc counterclockwise

Full-surface machining with opposing spindle

G30: The expert program switches the kinematics of the counterspindle. In addition, G30 activates the mirroring of the Z-axis and converts other functions (e.g. circular arcs G2, G3).

G99: The expert program shifts the contour and mirrors the coordinate system (Z axis). Further programming of G99 is normally not required for machining the rear face after rechucking.

Example: The workpiece is machined on the front face, transferred to the opposing spindle through an expert program and machined on the rear face (see illustrations).

The expert program is used for:

- Angle-synchronous workpiece transfer to the opposing spindle
- Mirroring traverse paths in the Z axis
- Activating a conversion list
- Mirroring the contour description and shifting for the 2nd chucking

Full-surface machining on machines with opposing spindles

HEADER	
#MATERIAL STEEL	
#UNIT METRIC	
TURRET	
T1 ID "512-600.10"	
T2 ID "111-80-080.1"	
T102 ID "115-80-080.1"	
BLANK	
N1 G20 X100 Z100 K1	
Finished part	
. . .	
FRONT Z0	
N 13 G308 ID"Line" P-1	
N 14 G100 XK-15 YK10	
N 15 G101 XK-10 YK12 BR2	
N 16 G101 XK-4.0725 YK-12.6555 BR4	
N 18 G101 XK10	
N 19 G309	
REAR SIDE Z-98	
. . .	
MACHINING	

N27 G59 Z233	Zero point shift for 1st setup
N28 G0 W#iS18	Counterspindle to machining position
N30 G14 Q0	
N31 G26 S2500	
N32 T2	
. . .	
N63 M5	
N64 T1	
N65 G197 S1485 G193 F0.05 M103	C-axis machining in the main spindle
N66 M14	
N67 M107	
N68 G0 X36.0555 Z3	
N69 G110 C146.31	
N70 G147 I2 K2	
N71 G840 Q0 NS15 NE18 I0.5 R0 P1	
N72 G0 X31.241 Z3	
N73 G14 Q0	
N74 M105 M109	
N76 M15	Deactivate C axis
N80 L"RECHUCK" V1 LA.. LB.. LC..	Expert prog. for part transfer with following functions: G720 Spindle synchronization G916 Traversing to a fixed stop G30 Switch the kinematics G99 Mirroring and shifting of the workpiece contour
N90 G59 Z222	Datum shift 2nd setup
. . .	
N91 G14 Q0	
N92 T102	
N93 G396 S220 G395 F0.2 M304	Technology data for opposing spindle
N94 M107	Turning in the counterspindle
N95 G0 X120 Z3	
N96 G810	Fixed cycles
N97 G30 Q0	Switch off rear-face machining
. . .	
N129 M30	
END	

Full-surface machining with single spindle

G30: Normally not required

G99: The expert program mirrors the contour. Further programming of G99 is normally not required for machining the rear face after rechucking.

Example: Describes the machining of the front and rear face, using **one** NC program. The workpiece is first machined on the front face; then it is rechucked manually. The rear face is machined subsequently.

The expert program mirrors and shifts the contour for the 2nd setup.

Full-surface machining on machine with one spindle

HEADER	
#MATERIAL STEEL	
#UNIT METRIC	
TURRET	
T1 ID "512-600.10"	
T2 ID "111-80-080.1"	
T4 ID "121-55-040.1"	
BLANK	
N1 G20 X100 Z100 K1	
Finished part	
. . .	
FRONT Z0	
. . .	
REAR SIDE Z-98	
N20 G308 ID"R" P-1	
N21 G100 XK5 YK-10	
N22 G101 YK15	
N23 G101 XK-5	
N24 G103 XK-8 YK3.8038 R6 I-5	
N25 G101 XK-12 YK-10	
N26 G309	
MACHINING	

N27 G59 Z233	Zero point shift for 1st setup
. . .	
N82 M15	Prepare the rechucking
N86 G99 H1 V0 K-98	Contour mirroring and shifting for manual rechucking
N87 M0	Stop for rechucking
N88 G59 Z222	Datum shift for 2nd setup
. . .	
N125 M5	Milling – rear face
N126 T1	
N127 G197 S1485 G193 F0.05 M103	
N128 M14	
N130 M107	
N131 G0 X22.3607 Z3	
N132 G110 C-116.565	
N134 G147 I2 K2	
N135 G840 Q0 NS22 NE25 I0.5 R0 P1	
N136 G0 X154 Z-95	
N137 G0 X154 Z3	
N138 G14 Q0	
N139 M105 M109	
N142 M15	
N143 G30 Q0	Switch off rear-face machining
N144 M30	
END	

5

Touch probe cycles

5.1 General information on touch probe cycles (software option)

The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe. The machine manual provides further information.

Please note that HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used!

Principle of function of touch probe cycles

When you run a touch probe cycle, the 3-D touch probe is pre-positioned at positioning feed rate. The actual probing movement is then executed from there at probing feed rate. The machine tool builder determines the positioning feed rate for the touch probe in a machine parameter. You define the probing feed rate in the respective touch probe cycle.

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to the starting position of the probing procedure at positioning feed rate.

If the stylus is not deflected within a defined distance, the control displays an error message.

Touch probe cycles for automatic operation

The control provides numerous touch probe cycles for various applications:

- Calibrating a touch trigger probe
- Measuring circles, circle segments, angle and position of the C axis
- Misalignment compensation
- Single- point and double-point measurement
- Finding a hole or stud
- Zero point setting in the Z or C axis
- Automatic tool measurement

Touch probe cycles are programmed via G functions in DIN PLUS. Just like the fixed cycles, also the touch probe cycles use transfer parameters.

To simplify programming, the TNC shows a graphic during cycle definition. The appropriate input parameters are displayed in the help graphic (see figure at right).

The touch probe cycles save status information and measuring results in the variable #i99. Depending on the input parameters in the touch probe cycle you can interrogate the following values:

Result #i99	Meaning
< 999997	Measuring result
999999	Touch probe not deflected
-999999	Invalid measuring axis programmed
999998	Maximum deviation **WE** exceeded
999997	Maximum compensation value **E** exceeded

Programming the the touch probe cycle in DIN PLUS

ISO Mode

- Select DIN PLUS programming and place the cursor in the MACHINING program section.
- Select "Machining" pull-down menus
- Select "G menu" pull-down menus
- Select "Touch probe cycles" pull-down menus
- Select measuring cycle group
- Select the cycle

Group of measuring cycles	Page
Single-point measurements	Page 336
Double-point measurements	Page 358
Calibration cycles	Page 412
Probing	Page 462
Search cycles	Page 478
Circle measurement	Page 510
Angle position	Page 510
In-process measurement	Page 510

Example: Touch probe cycle in the DINplus program

```
HEADER
#MATERIAL          Steel
#UNIT              METRIC

TURRET 1
T1 ID"342-300.1"
T2 ID"111-80-080.1"
. . .

BLANK
N1 G20 X120 Z120 K2

FINISHED PART
N2 G0 X60 Z-115
N3 G1 Z-105
. . .
MACHINING
N19 T1
N19 G0 X0 Z5
N20 G771 R1 D0 K-30 AC0 BD2 Q0 P0 H0
N1 T2 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-25 A5 V2 [drilling]
. . .
END
```

5.2 Touch probe cycles for single-point measurement

Single-point measurement for tool compensation G770

Cycle G770 measures with the programmed measuring axis in the specified direction. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of compensation:

- 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
- 2: Recessing tool **Dx**/**DS**
- 4: Milling tool **DD**

D Measuring axis: Axis in which the measurement is to be made

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

AC Nominal value for target position: Touch point coordinate

BD Tolerance +/-: Measurement result range in which no compensation is applied

WT Compensation number **T** or **G149**:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

E Maximum compensation value for the tool compensation

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

Example: G770—Single-point measurement for tool compensation

```
. . .
MACHINING
N3 G770 R1 D0 K20 AC0 BD0.2 WT3 V1 O1 Q0
P0 H0
. . .
```

Parameters

V Retraction type

- 0: Without: Only position touch probe back to the starting point if the touch probe was deflected
- 1: Automatic: Always position touch probe back to the starting point

O Error evaluation

- 0: Program: Do not interrupt program run, no error message
- 1: Automatic: Interrupt program run and output error message if touch probe is not deflected within measuring path

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Single-point measurement for zero point G771

Cycle G771 measures with the programmed measuring axis in the specified direction. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as zero point shift. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 1: Table and G59: Activate zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G59 for the further program run. Zero point shift no longer active after program run.

D Measuring axis: Axis in which the measurement is to be made

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

AC Nominal value for target position: Touch point coordinate

BD Tolerance +/-: Measurement result range in which no compensation is applied

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Example: G771 Single-point measurement for tool compensation

```
. . .
MACHINING
N3 G771 R1 D0 K20 AC0 BD0.2 Q0 P0 H0
. . .
```

Parameters

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Zero point C axis, single-point measurement G772

Cycle G772 measures with the C axis in the specified direction. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as zero point shift. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position, the element to be probed is moved toward the touch probe by a rotation of the C axis. When the workpiece touches the stylus, the measured value is saved and the workpiece is returned.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 1: Table and G152: Activate zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G152 for the further program run. Zero point shift no longer active after program run.

C Incremental measuring path with direction: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.

AC Nominal value for target position: Absolute coordinate of touch point in degrees

BD Tolerance +/-: Measurement result range (in degrees) in which no compensation is applied

KC Compensation offset: Additional compensation value that is applied to the zero point result

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Example: G772—Single-point measurement zero point C axis

```
. . .
MACHINING
N3 G772 R1 C20 AC0 BD0.2 Q0 P0 H0
. . .
```

Parameters

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Zero point C-axis object center G773

Cycle G773 measures an element with the C axis from two opposite sides and places the center of the element to a defined position. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position, the element to be probed is moved toward the touch probe by a rotation of the C axis. When the workpiece touches the stylus, the measured value is saved and the workpiece is returned. Then the touch probe is pre-positioned for the opposite probing procedure. When the second measured value has been determined, the cycle computes the mean value of the two measurements and applies a zero point shift in the C axis. The nominal position **AC** defined in the cycle is then in the center of the probed element.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, each measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 1: Table and G152: Activate zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G152 for the further program run. Zero point shift no longer active after program run.

C Incremental measuring path with direction: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.

E Circumnavigation axis: Axis that is positioned back by RB in order to circumnavigate the element

RB Circumnavigation direction offset: Retraction value in the circumnavigation axis **E** for pre-positioning for the next probing position

RC C-angle offset: Difference in the C axis between the first and the second measuring position

AC Nominal value for target position: Absolute coordinate of touch point in degrees

BD Tolerance +/-: Measurement result range (in degrees) in which no compensation is applied

KC Compensation offset: Additional compensation value that is applied to the zero point result

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Example: G773—Single-point measurement C-axis object center

```
. . .
MACHINING
N3 G773 R1 C20 E0 RB20 RC45 AC30 BD0.2 Q0
P0 H0
. . .
```

Parameters

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

5.3 Touch probe cycles for two-point measurement

Two-point measurement G18 transverse G775

Cycle G775 measures two opposite points in the **X/Z plane** with the **measuring axis X**. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the offset in the circumnavigation direction **RB** and then by the offset in the measuring direction **RC**. The cycle executes the second probing operation in the opposite direction, saves the result and positions the touch probe back with the circumnavigation axis by the circumnavigation value.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring points are approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of compensation:

- 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
- 2: Recessing tool **Dx**/**DS**
- 3: Cutter **DX/DD**
- 4: Milling cutter **DD**

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

E Circumnavigation axis: Selection of axis for retraction movement between the probing positions:

- 0: Z axis
- 2: Y axis

RB Circumnavigation direction offset: Distance

RC Offset X: Distance for pre-positioning before the second measurement

XE Nominal value for target position X: Absolute coordinate of touch point

BD Tolerance +/-: Range for the first measurement result in which no compensation is applied

Example: G775—Two-point measurement for tool compensation

```
. . .
MACHINING
N3 G775 R1 K20 E1 XE30 BD0.2 X40 BE0.3
WT5 Q0 P0 H0
. . .
```

Parameters

X Nominal width X: Coordinate for the second probing position

BE Tolerance width +/-: Range for the second measurement result in which no compensation is applied

WT Compensation number **T** or **G149**, first measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

AT Compensation number **T** or **G149**, second measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

FP Maximum permissible compensation

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

The cycle computes the compensation value **WT** from the result of the first measurement and the compensation value **AT** from the result of the second measurement.

Two-point measurement G18 longitudinal G776

Cycle G776 measures two opposite points in the **X/Z plane** with the **measuring axis Z**. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the offset in the circumnavigation direction **RB** and then by the offset in the measuring direction **RC**. The cycle executes the second probing operation in the opposite direction, saves the result and positions the touch probe with the circumnavigation axis by the circumnavigation value.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring points are approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of compensation:

- 1: Tool compensation **DX/DZ** for turning tool or additive compensation
- 2: Recessing tool **Dx/DS**
- 3: Cutter **DX/DD**
- 4: Milling cutter **DD**

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

E Circumnavigation axis: Selection of axis for retraction movement between the probing positions:

- 0: X axis
- 2: Y axis

RB Circumnavigation direction offset: Distance

RC Offset Z: Distance for pre-positioning before the second measurement

ZE Nominal value for target position Z: Absolute coordinate of touch point

BD Tolerance +/-: Range for the first measurement result in which no compensation is applied

Z Nominal width Z: Coordinate for the second probing position

BE Tolerance width +/-: Range for the second measurement result in which no compensation is applied

Example: G776—Two-point measurement for tool compensation

```
. . .
MACHINING
N3 G775 R1 K20 E1 XE30 BD0.2 X40 BE0.3
WT5 Q0 P0 H0
. . .
```

Parameters

WT Compensation number **T** or **G149**, first measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

AT Compensation number **T** or **G149**, second measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

FP Maximum permissible compensation

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

The cycle computes the compensation value **WT** from the result of the first measurement and the compensation value **AT** from the result of the second measurement.

Two-point measurement G17 longitudinal G777

Cycle G777 measures two opposite points in the **X/Y plane** with the **measuring axis Y**. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the offset in the circumnavigation direction **RB** and then by the offset in the measuring direction **RC**. The cycle executes the second probing operation in the opposite direction, saves the result and positions the touch probe with the circumnavigation axis by the circumnavigation value.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring points are approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of compensation:

- 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
- 2: Recessing tool **Dx**/**DS**
- 3: Milling tool**DX/DD**
- 4: Milling tool **DD**

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

RB Circumnavigation direction offset: Distance in circumnavigation direction X

RC Offset Z: Distance for pre-positioning before the second measurement

YE Nominal value for target position Y: Absolute coordinate of touch point

BD Tolerance +/-: Range for the first measurement result in which no compensation is applied

Y Nominal width Z: Coordinate for the second probing position

BE Tolerance width +/-: Range for the second measurement result in which no compensation is applied

Example: G777—Two-point measurement for tool compensation

```
. . .
MACHINING
N3 G777 R1 K20 YE10 BD0.2 Y40 BE0.3 WT5
Q0 P0 H0
. . .
```

Parameters

WT Compensation number **T** or **G149**, first measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

AT Compensation number **T** or **G149**, second measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

FP Maximum permissible compensation

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

The cycle computes the compensation value **WT** from the result of the first measurement and the compensation value **AT** from the result of the second measurement.

Two-point measurement G19 longitudinal G778

Cycle G778 measures two opposite points in the **Y/Z plane** with the **measuring axis Y**. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the offset in the circumnavigation direction **RB** and then by the offset in the measuring direction **RC**. The cycle executes the second probing operation in the opposite direction, saves the result and positions the touch probe back with the circumnavigation axis by the circumnavigation value.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring points are approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of compensation:

- 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
- 2: Recessing tool **Dx**/**DS**
- 3: Milling tool**DX/DD**
- 4: Milling tool **DD**

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

RB Circumnavigation direction offset: Distance in circumnavigation direction X

RC Offset Y: Distance for pre-positioning before the second measurement

ZE Nominal value for target position Y: Absolute coordinate of touch point

BD Tolerance +/-: Range for the first measurement result in which no compensation is applied

Z Nominal width Y: Coordinate for the second probing position

BE Tolerance width +/-: Range for the second measurement result in which no compensation is applied

Example: G778—Two-point measurement for tool compensation

```
. . .
MACHINING
N3 G778 R1 K20 YE30 BD0.2 Y40 BE0.3 WT5
Q0 P0 H0
. . .
```

Parameters

WT Compensation number **T** or **G149**, first measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

AT Compensation number **T** or **G149**, second measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

FP Maximum permissible compensation

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

> The cycle computes the compensation value **WT** from the result of the first measurement and the compensation value **AT** from the result of the second measurement.

5.4 Calibrating touch probes

Calibrate touch probe standard G747

Cycle G747 measures with the programmed axis and, depending on the selected calibration method, calculates the touch probe adjustment dimension or the ball diameter. If the tolerance values defined in the cycle are exceeded, the cycle corrects the touch probe data. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Calibration method:
- 0: Change ball diameter
- 1: Change adjustment dimension

D Measuring axis: Axis in which the measurement is to be made

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

AC Nominal value for target position: Touch point coordinate

BD Tolerance +/-: Measurement result range in which no compensation is applied

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Example: G747 Calibrate touch probe

```
. . .
MACHINING
N3 G747 R1 K20 AC10 BD0.2 Q0 P0 H0
. . .
```

Parameters

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Calibrate touch probe via two points G748

Cycle G748 measures two opposite points and computes the touch probe adjustment dimension and the ball diameter. If the tolerance values defined in the cycle are exceeded, the cycle corrects the touch probe data. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the offset in the circumnavigation direction **RB** and then by the offset in the measuring direction **RC**. The cycle performs the second probing operation in the opposite direction and saves the result.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring points are approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

RB Circumnavigation direction offset: Distance

RC Measuring direction offset: Distance for pre-positioning before the second measurement

AC Nominal value for target position: Absolute touch point coordinate

EC Nominal width: Coordinate for the second probing position

BE Tolerance width +/-: Range for the second measurement result in which no compensation is applied

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

Example: G748 Calibrate touch probe via two points

```
. . .
MACHINING
N3 G748 K20 AC10 EC33 Q0 P0 H0
. . .
```

Parameters

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

5.5 Measuring with touch probe cycles

Paraxial probing G764

Cycle G764 measures with the programmed axis and displays the measured values on the control screen. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters

D Measuring axis: Axis in which the measurement is to be made

K Incremental measuring path with direction (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

V Retraction type

- 0: Without: Only position touch probe back to the starting point if the touch probe was deflected
- 1: Automatic: Always position touch probe back to the starting point

O Error evaluation

- 0: Program: Do not interrupt program run, no error message
- 1: Automatic: Interrupt program run and output error message if touch probe is not deflected within measuring path

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

Example: G764 paraxial probing

```
. . .
MACHINING
N3 G764 D0 K20 V1 O1 Q0 P0 H0
. . .
```

Probing in C axis G765

Cycle G765 measures with the C axis and displays the measured values on the control screen. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position, the element to be probed is moved toward the touch probe by a rotation of the C axis. When the workpiece touches the stylus, the measured value is saved and the workpiece is returned.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters

C Incremental measuring path with direction: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.

V Retraction type

- 0: Without: Only position touch probe back to the starting point if the touch probe was deflected
- 1: Automatic: Always position touch probe back to the starting point

O Error evaluation

- 0: Program: Do not interrupt program run, no error message
- 1: Automatic: Interrupt program run and output error message if touch probe is not deflected within measuring path

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

Example: G765 Probing in C axis

```
. . .
MACHINING
N3 G765 C20 V1 O1 AC0 BD0.2 Q0 P0 H0
. . .
```

Probing in two axes G766

Cycle G765 measures the position programmed in the cycle in the **X/Z plane** and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measuring results should be saved.

Cycle run

The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters

Z Target point Z: Z coordinate of measuring point

X Target point X: X coordinate of measuring point

V Retraction type

- 0: Without: Only position touch probe back to the starting point if the touch probe was deflected
- 1: Automatic: Always position touch probe back to the starting point

O Error evaluation

- 0: Program: Do not interrupt program run, no error message
- 1: Automatic: Interrupt program run and output error message if touch probe is not deflected within measuring path

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

Example: G766 Probing in two axes in X/Z plane

```
. . .
MACHINING
N3 G766 Z-5 X30 V1 O1 AC0 BD0.2 Q0 P0 H0
. . .
```

Probing in two axes G768

Cycle G765 measures the position programmed in the cycle in the **Z/Y plane** and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measuring results should be saved.

Cycle run

The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters

Z Target point Z: Z coordinate of measuring point

Y Target point Y: Y coordinate of measuring point

V Retraction type
- 0: Without: Only position touch probe back to the starting point if the touch probe was deflected
- 1: Automatic: Always position touch probe back to the starting point

O Error evaluation
- 0: Program: Do not interrupt program run, no error message
- 1: Automatic: Interrupt program run and output error message if touch probe is not deflected within measuring path

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measurement result is saved automatically under the next consecutive number.

P PRINT outputs
- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement
- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

Example: G768 Probing in two axes in Z/Y plane

```
. . .
MACHINING
N3 G768 Z-5 Y10 V1 O1 AC0 BD0.2 Q0 P0 H0
. . .
```

Probing in two axes G769

Cycle G765 measures the position programmed in the cycle in the **X/Y plane** and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measurement results should be saved.

Cycle run

The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters

X Target point X: X coordinate of measuring point

Y Target point Y: Y coordinate of measuring point

V Retraction type

- 0: Without: Only position touch probe back to the starting point if the touch probe was deflected
- 1: Automatic: Always position touch probe back to the starting point

O Error evaluation

- 0: Program: Do not interrupt program run, no error message
- 1: Automatic: Interrupt program run and output error message if touch probe is not deflected within measuring path

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measurement result is saved automatically under the next consecutive number.

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

Example: G769 Probing in two axes in X/Y plane

```
. . .
MACHINING
N3 G769 X25 Y10 V1 O1 AC0 BD0.2 Q0 P0 H0
. . .
```

5.6 Search cycles

Find hole in C face G780

Cycle G780 probes the workpiece face several times with the Z axis. Prior to each probing, the touch probe is shifted by a distance defined in the cycle until a hole is found. Optionally, the cycle determines the mean value by two probing operations in the hole.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as zero point shift. The result of the measurement is saved additionally in the variable #i99.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in Maximum Deviation parameter **WE**.

Cycle run

From the current position the touch probe moves along the measuring axis **Z** toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. Then the cycle rotates the C axis by the angle defined in the Search Grid parameter **RC** and probes again with the Z axis. This process is repeated until a hole is found. In the hole the cycle performs two probing operations with the C axis, calculates the center of the hole and places the zero point in the C axis.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 1: Activate table and G152 zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G152 for the further program run. Zero point shift no longer active after program run.

D Result:

- 1: Position: Set zero point without determining the hole center. No probing operation in the hole.
- 2: Object center: Before the zero point is set, determine hole center in two probing operations with the C axis

K Incremental measuring path Z (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

C Starting position C: Position of the C axis for the first probing operation

Example: Find hole in C face G780

```
. . .
MACHINING
N3 G780 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0
P0 H0
. . .
```

Parameters

RC Search grid Ci: Stepping angle of the C axis for the subsequent probing operations

A Number of points: Maximum number of probing operations

IC Measuring path C: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.

AC Nominal value for target position: Absolute coordinate of touch point in degrees

BD Tolerance +/-: Measurement result range (in degrees) in which no compensation is applied

KC Compensation offset: Additional compensation value that is applied to the zero point result

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measurement result is saved automatically under the next consecutive number.

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Find hole in C lateral surface G781

Cycle G780 probes the lateral surface of a workpiece several times with the X axis. Prior to each probing, the C axis is rotated by a distance defined in the cycle until a hole is found. Optionally, the cycle determines the mean value by two probing operations in the hole.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as zero point shift. The result of the measurement is saved additionally in the variable #i99.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in Maximum Deviation parameter **WE**.

Cycle run

From the current position the touch probe moves along the measuring axis **X** toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. Then the cycle rotates the C axis by the angle defined in the Search Grid parameter **RC** and probes again with the X axis. This process is repeated until a hole is found. In the hole the cycle performs two probing operations with the C axis, calculates the center of the hole and places the zero point in the C axis.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 1: Activate table and G152 zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G152 for the further program run. Zero point shift no longer active after program run.

D Result:

- 1: Position: Set zero point without determining the hole center. No probing operation in the hole.
- 2: Object center: Before the zero point is set, determine hole center in two probing operations with the C axis

K Incremental measuring path X (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

C Starting position C: Position of the C axis for the first probing operation

RC Search grid Ci: Stepping angle of the C axis for the subsequent probing operations

A Number of points: Maximum number of probing operations

Example: G781 Find hole in C face

```
. . .
MACHINING
N3 G781 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0
P0 H0
. . .
```

Parameters

IC Measuring path C: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.

AC Nominal value for target position: Absolute coordinate of touch point in degrees

BD Tolerance +/-: Measurement result range (in degrees) in which no compensation is applied

KC Compensation offset: Additional compensation value that is applied to the zero point result

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Find stud in C face G782

Cycle G782 probes the workpiece face several times with the Z axis. Prior to each probing, the C axis is rotated by a distance defined in the cycle until a stud is found. Optionally, the cycle determines the mean value by two probing operations on the stud diameter.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as zero point shift. The result of the measurement is saved additionally in the variable #i99.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in Maximum Deviation parameter **WE**.

Cycle run

From the current position the touch probe moves along the measuring axis **X** toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. Then the cycle rotates the C axis by the angle defined in the Search Grid parameter **RC** and probes again with the X axis. This process is repeated until a stud is found. The cycle performs two probing operations on the stud diameter with the C axis, calculates the center of the hole and places the zero point in the C axis.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:
- 1: Activate table and G152 zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G152 for the further program run. Zero point shift no longer active after program run.

D Result:
- 1: Position: Set zero point without determining the stud center. The stud diameter is not probed.
- 2: Object center: Before the zero point is set, determine stud center in two probing operations with the C axis.

K Incremental measuring path X (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

C Starting position C: Position of the C axis for the first probing operation

RC Search grid Ci: Stepping angle of the C axis for the subsequent probing operations

Example: G782 Find stud in C face

```
. . .
MACHINING
N3 G782 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0
P0 H0
. . .
```

Parameters

- A Number of points: Maximum number of probing operations
- IC Measuring path C: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.
- AC Nominal value for target position: Absolute coordinate of touch point in degrees
- BD Tolerance +/-: Measurement result range (in degrees) in which no compensation is applied
- KC Compensation offset: Additional compensation value that is applied to the zero point result
- WE Maximum deviation: Probe twice and monitor the dispersion of the measured values
- F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.
- Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)
- P PRINT outputs
  - 0: OFF: Do not show measuring results
  - 1: ON: Display measuring results
- H INPUT instead of measurement
  - 0: Default: Obtain measured values by probing
  - 1: PC test: Simulate probing cycle on the programming station
- AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Find stud in C lateral surface G783

Cycle G783 probes the workpiece face several times with the X axis. Prior to each probing, the touch probe is shifted by a distance defined in the cycle until a stud is found. Optionally, the cycle determines the mean value by two probing operations on the stud diameter.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as zero point shift. The result of the measurement is saved additionally in the variable #i99.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in Maximum Deviation parameter **WE**.

Cycle run

From the current position the touch probe moves along the measuring axis **Z** toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. Then the cycle rotates the C axis by the angle defined in the Search Grid parameter **RC** and probes again with the Z axis. This process is repeated until a stud is found. The cycle performs two probing operations on the stud diameter with the C axis, calculates the center of the hole and places the zero point in the C axis.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 1: Activate table and G152 zero point shift and additionally save in zero point table. The zero-point shift also remains active after the program run.
- 2: Activate zero point shift with G152 for the further program run. Zero point shift no longer active after program run.

D Result:

- 1: Position: Set zero point without determining the stud center. The stud diameter is not probed.
- 2: Object center: Before the zero point is set, determine stud center in two probing operations with the C axis.

K Incremental measuring path Z (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

C Starting position C: Position of the C axis for the first probing operation

RC Search grid Ci: Stepping angle of the C axis for the subsequent probing operations

A Number of points: Maximum number of probing operations

Example: G783 Find stud in C lateral surface

```
. . .
MACHINING
N3 G783 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0
P0 H0
. . .
```

Parameters

IC Measuring path C: Measuring path of the C axis (in degrees), starting from the current position. The algebraic sign determines the probing direction.

AC Nominal value for target position: Absolute coordinate of touch point in degrees

BD Tolerance +/-: Measurement result range (in degrees) in which no compensation is applied

KC Compensation offset: Additional compensation value that is applied to the zero point result

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

5.7 Circular measurement

Circular measurement G785

Cycle G785 determines the circle center and diameter by probing three times in the programmed plane and shows the measured values on the control screen. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves in the defined measuring plane toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. Another two probing operations are carried out with the defined stepping angle. If a starting diameter **D** was programmed, the cycle positions the touch probe on a circular path before the respective measuring process.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:
- 0: X/Y plane G17: Probe circle in X/Y plane
- 1: Z/X plane G18: Probe circle in Z/X plane
- 2: Y/Z plane G19: Probe circle in Y/Z plane

BR Inside/outside:
- 0: Inside: Probe inside diameter
- 1: Outside: Probe outside diameter

K Incremental measuring path (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

C Angle of 1st measurement: Angle for the first probing operation

RC Incremental angle: Stepping angle for the subsequent probing operations

D Starting diameter: Diameter on which the touch probe is pre-positioned before the measurements.

WB Position in infeed direction: Measuring height to which the touch probe is positioned before the measuring process. No input: The circle is probed from the current position.

I Circle center in axis 1: Nominal position of the circle center in first axis

J Circle center in axis 2: Nominal position of the circle center in second axis

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

Example: G785 Circular measurement

```
. . .
MACHINING
N3 G785 R0 BR0 K2 C0 RC60 I0 J0 Q0 P0 H0
. . .
```

Parameters

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measurement result is saved automatically under the next consecutive number.

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Determine pitch circle G786

Cycle G786 determines the center and diameter of a pitch circle by measuring three holes and shows the measured values on the control screen. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves in the defined measuring plane toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point. Another two probing operations are carried out with the defined stepping angle. If a starting diameter **D** was programmed, the cycle positions the touch probe on a circular path before the respective measuring process.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Type of zero point shift:

- 0: X/Y plane G17: Probe circle in X/Y plane
- 1: Z/X plane G18: Probe circle in Z/X plane
- 2: Y/Z plane G19: Probe circle in Y/Z plane

K Incremental measuring path: Maximum measuring path for measurement in the holes.

C Angle of 1st hole: Angle for the first probing operation

AC Angle of 2nd hole: Angle for the second probing operation

RC Angle of 3rd hole: Angle for the third probing operation

WB Position in infeed direction: Measuring height to which the touch probe is positioned before the measuring process. No input: The hole is probed from the current position.

I Pitch circle center in axis 1: Nominal position of the pitch circle center in first axis

J Pitch circle center in axis 2: Nominal position of the pitch circle center in second axis

D Nominal diameter: Diameter on which the touch probe is pre-positioned before the measurements.

WS Max. diameter of pitch circle

WC Min. diameter of pitch circle

BD Tolerance for center in first axis

BE Tolerance for center in second axis

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

Example: G786 Determine pitch circle

```
. . .
MACHINING
N3 G786 R0 K8 I0 J0 D50 WS50.1 WC49.9
BD0.1 BE0.1 P0 H0
. . .
```

Parameters

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measuring result is saved automatically under the next consecutive number.

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

5.8 Measure angle

Angular measurement G787

Cycle G787 probes twice in the programmed direction and computes the angle. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation for a subsequent misalignment compensation. Program Cycle G788 next in order to activate the misalignment compensation. The result of the measurement is saved additionally in the variable #i99 (See "Touch probe cycles for automatic operation" on page 431.).

Cycle run

From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is retracted. Then the touch probe is pre-positioned for the second measurement and the workpiece is probed.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path. If a maximum deviation **WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the maximum deviation **WE**, the program run is interrupted and an error message is displayed.

Parameters

R Evaluation:

- 1: Prepare tool compensation and misalignment compensation:
- 2: Prepare misalignment compensation
- 3: Angle output:

D Directions:

- 0: X measurement, Z offset
- 1: Y meas. Z offset
- 2: Z measurement, X offset
- 3: Y measurement, X offset
- 4: Z measurement, Y offset
- 5: X measurement, Y offset

K Incremental measuring path (signed): Maximum measuring path for probing. The algebraic sign determines the probing direction.

WS Position of first measuring point

WC Position of second measuring point

AC Nominal angle of measured surface

BE Angle tolerance +/-: Measurement result range (in degrees) in which no compensation is applied

RC Target position of first measurement: Nominal value of first measuring point

BD Tolerance of first measurement +/-: Range for the measurement result in which no compensation is applied

Example: G787 Angular measurement

```
. . .
MACHINING
N3 G787 R1 D0 BR0 K2 WS-2 WC15 AC170 BE1
RC0 BD0.2 WT3 Q0 P0 H0
. . .
```

Parameters

WT Compensation number **T** or **G149**, first measured edge:

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation D9xx to correct the difference to the nominal value (only possible with compensation type **R** =1)

FP Maximum permissible compensation

WE Maximum deviation: Probe twice and monitor the dispersion of the measured values

F Measuring feed rate: Feed rate for probing. If nothing is entered, the measuring feed rate from the touch probe table is used. If the entered measuring feed rate **F** is higher than the one in the touch probe table, the feed rate is reduced to the value from the touch probe table.

Q Tool orientation: Orient the touch probe in the programmed probing direction before each probing operation (machine-dependent function)

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measuring result is saved automatically under the next consecutive number.

P PRINT outputs

- 0: OFF: Do not show measuring results
- 1: ON: Display measuring results

H INPUT instead of measurement

- 0: Default: Obtain measured values by probing
- 1: PC test: Simulate probing cycle on the programming station

AN Log no.: Save measurement results in "TNC:\table\messpro.mep" table (line numbers 0–99; the table can be expanded if necessary)

Misalignment compensation after angle measurement G788

Cycle G788 activates a misalignment compensation determined with Cycle G787 Angle Measurement.

Parameters

NF Result variable no.: Number of the first global variable in which the result is saved (no entry = variable 810). The second measurement result is saved automatically under the next consecutive number.

P Compensation

- 0: OFF: Do not perform misalignment compensation
- 1: ON: Perform misalignment compensation

Example: G788 Misalignment compensation after angle measurement

```
. . .
MACHINING
N3 G788 NF1 P0
. . .
```

5.9 In-process measurement

Measure workpieces (option)

In-process measurement is measurement at the workpiece with a touch probe located in a tool holder of the machine. In the tool list, enter your touch probe as a new tool. Use the tool type "touch probe." The following cycles for "in-process measurement" are basic cycles for probing functions that you can use to program individually adapted probing sequences.

Switch on measurement G910

G910 activates the selected touch probe.

Parameters

H Measuring direction (no function)

V Type of measurement

- 0: Touch probe (for workpiece measurement)
- 1: Table-mounted touch probe (for tool measurement)

Example: In-process measurement

```
. . .
N1 G0 X105 Z-20
N2 G94 F500
N3 G910 H0 V0
N4 G911 V0
N4 G1 Xi-10
N5 G914
N4 G912 Q1
N4 G913
N4 G0 X115
N4 #l1=#a9(X,0)
N4 IF NDEF(#l1)
N4 THEN
N4   PRINT("Probe not reached")
N4 ELSE
N4   PRINT ("Result of measurement:",#l1)
N4 ENDIF
. . .
```

Measuring path monitoring G911

G911 activates the measuring path monitoring. Then only a single feed path is permissible.

Parameters

V
- 0: Axes stay stationary with deflected touch probe
- 1: Axes automatically retract after deflection of the touch probe

Measured value capture G912

G912 puts the positions at which the touch probe was deflected into the result variables.

Parameters

Q Error evaluation when the touch probe is not reached

- 0: Error message of NC, program stops
- 1: Error evaluation in the NC program, measuring results="NDEF"

The measurement results are available in the following variables:

#a9(axis,channel)
Axis=axis name
Channel=channel number, 0=current channel

Example: Measurement results:

```
. . .
N1 #I1=#a9(X,0) [X value of current channel]
N2 #I2=#a9(Z,1) [Z value of channel 1 ]
N3 #I3=#a9(Y,0) [Y value of current channel]
N4 #I4=#a9(C,0) [C value of current channel]
. . .
```

End in-process measuring G913

G913 ends the measuring process.

Switch off measuring-path monitoring G914

G914 deactivates the measuring-path monitoring.

In-process measurement example: Measuring and compensating workpieces

The Control provides subprograms for the measurement of workpieces:

- measure_pos.ncs (German dialog texts)
- measure_pos_e.ncs (English dialog texts)

The programs require a touch probe as a tool. Beginning from the current position or the defined starting position, the Control moves along a measuring path in the entered axis direction. At the end it returns again to the previous position. The result of measurement can be included in error compensation.

The following subprograms are used:

- measure_pos_move.ncs
- _Print_txt_lang.ncs

Parameters

LA Measurement starting point in X (diameter value)—no input, current position.

LB Measurement starting point in Z (no input = current position).

LC Type of approach to measurement starting point

- 0: Diagonal
- 1: First X, then Z
- 2: First Z, then X

LD Measuring axis

- 0: X axis
- 1: Z axis
- 2: Y axis

LE Incremental measurement path. The algebraic sign defines the direction of traverse.

LF Measuring feed rate in mm/min—no input, the measuring feed rate from the touch probe table is used.

LH Nominal value of the target position

LI Tolerance +/-. If the measured deviation lies within this tolerance, the entered compensation value is not changed.

LJ 1: The measurement result is output as PRINT.

LK Number of the compensation value to be changed.

- 1-xx Turret pocket number of the tool to be compensated
- 901-916 Additive compensation
- Current tool number for touch probe calibration

LO Number of measurements:

- >0: The measurements are evenly distributed on the circumference with M19.
- <0: The measurements are made at the same position.

Parameters

LP Maximum permissible difference between the measurement results at a position. The program stops if the limit is violated.

LR Maximum permissible compensation value, <10 mm

LS 1: Program runs on the PC. Measurement results are interrogated through INPUT. For test purposes.

Example of in-process measurement: Measuring and compensating workpieces measure_pos_move.ncs

For the program "measure_pos_move.ncs" you have to use a touch probe as tool. The control moves the touch probe from the actual position in the given axis direction. After reaching the contact position, it returns again to the previous position. The measurement result can then be used again.

Parameters

LA Measuring axis:

- 0: X axis
- 1: Z axis
- 2: Y axis
- 3: C axis

LB Incremental measurement path. The algebraic sign defines the direction of traverse.

LC Measuring feed rate in mm/min

LD Retraction type

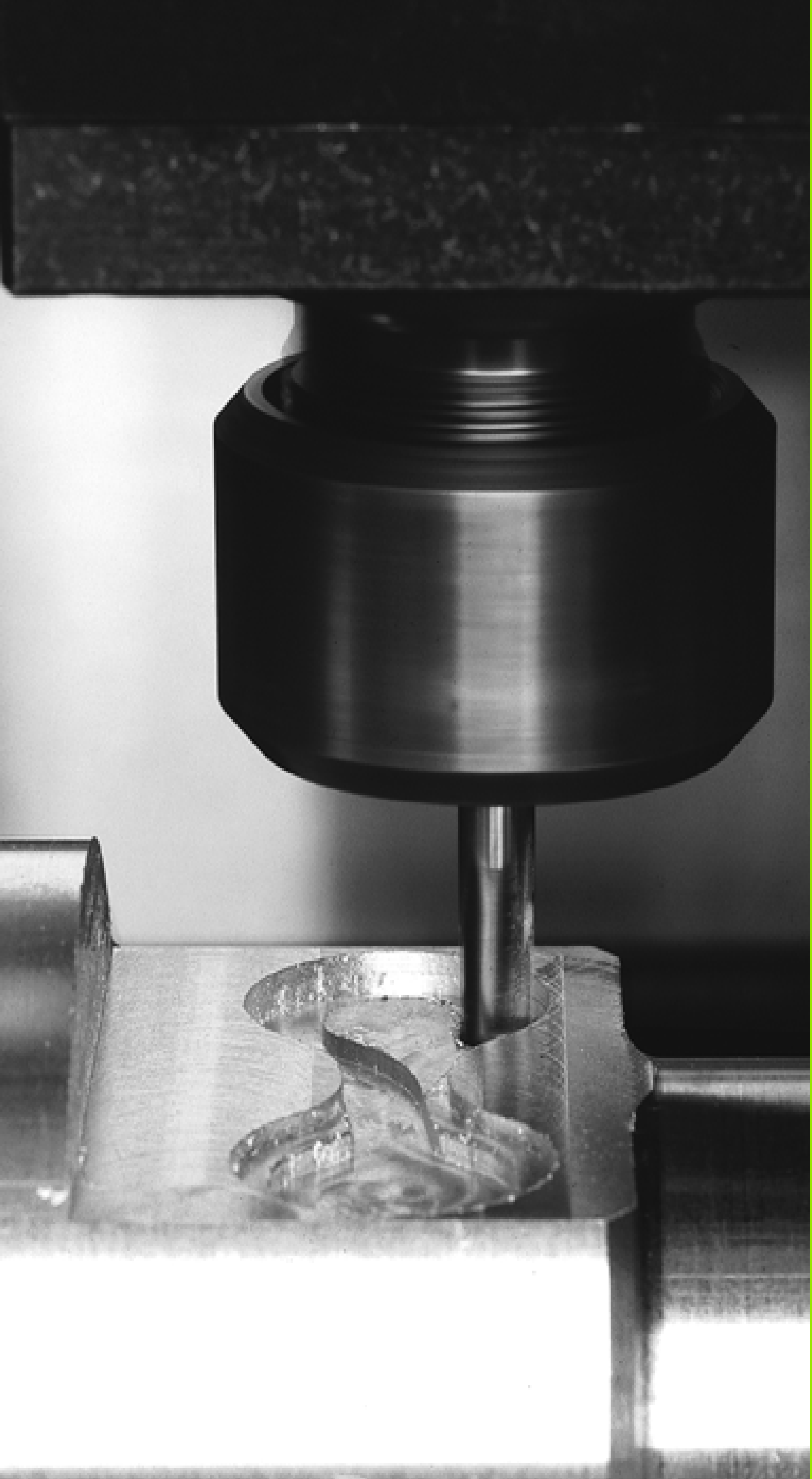
- 0: Return with G0 to starting point
- 1: Return automatically to starting point

LO Error response from missing stylus deflection:

- 0: A PRINT output follows and the program does not stop. A further response in the program is possible.
- 1: The programs stops with an NC error message.

LF 1: The measurement result is output as PRINT.

LS 1: Program runs on the PC. Measurement results are interrogated through INPUT. For test purposes.



6

DIN programming for the Y axis

6.1 Y-axis contours—Fundamentals

Position of milling contours

Define the reference plane or the reference diameter in the section code. Specify the depth and position of a milling contour (pocket, island) in the contour definition:

- With **depth P** programmed in the previous G308 cycle.
- Alternatively on figures: Cycle parameter **depth P**.

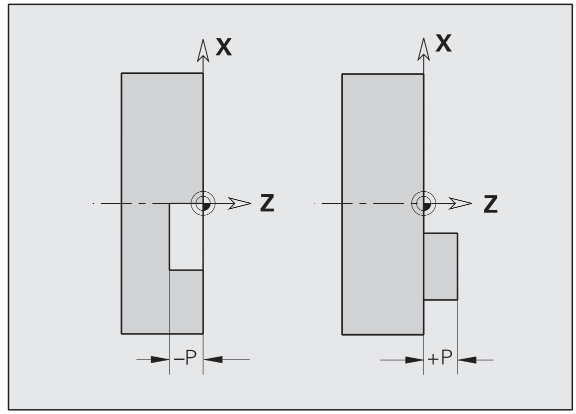
The **algebraic sign of "P"** defines the position of the milling contour:

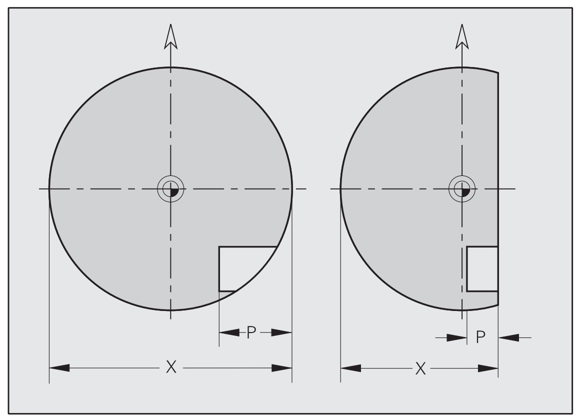
- P<0: Pocket
- P>0: Island

Position of milling contour Section	P	Surface	Milling floor
FRONT	P<0	Z	Z+P
	P>0	Z+P	Z
REAR SIDE	P<0	Z	Z–P
	P>0	Z–P	Z
SURFACE	P<0	X	X+(P*2)
	P>0	X+(P*2)	X

- X: Reference diameter from the section code
- Z: Reference plane from the section code
- P: Depth from G308 or from the figure definition

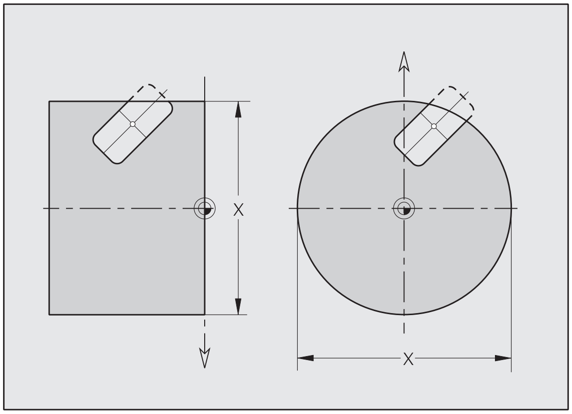
The area milling cycles mill the surface specified in the contour definition. **Islands** within this surface are not taken into consideration.





Cutting limit

If parts of the milling contour lie outside of the turning contour, you must limit the machining area with the **area diameter X** / **reference diameter X** (parameters of the section code or of the figure definition).

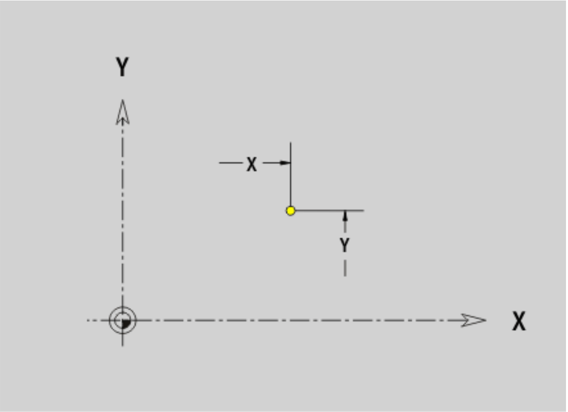


6.2 Contours in the XY plane

Starting point of contour in XY plane G170 Geo

G170 defines the starting point of a contour in the XY plane.

Parameters
X Starting point of contour (radius)
Y Starting point of contour



Line segment in XY plane G171 Geo

G171 defines a line segment in a contour of the XY plane.

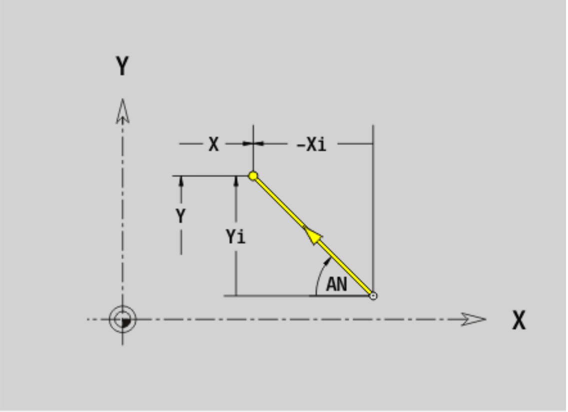
Parameters
X End point (radius)
Y End point
AN Angle to X axis (for direction of angle, see help graphic)
Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

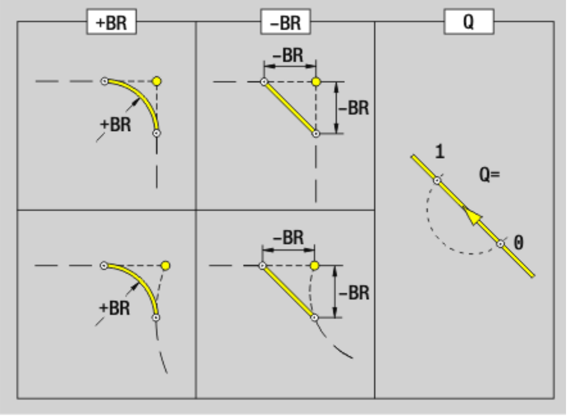
- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

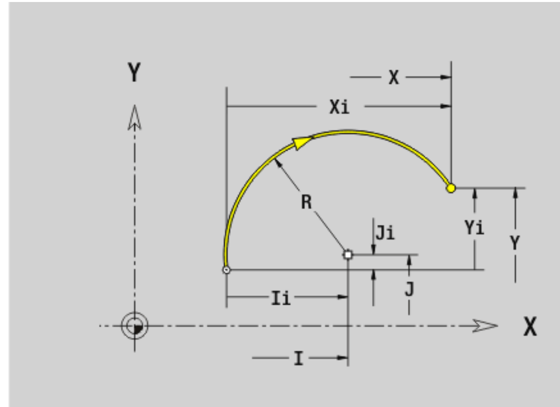
Programming X, Y: Absolute, incremental, modal or "?"

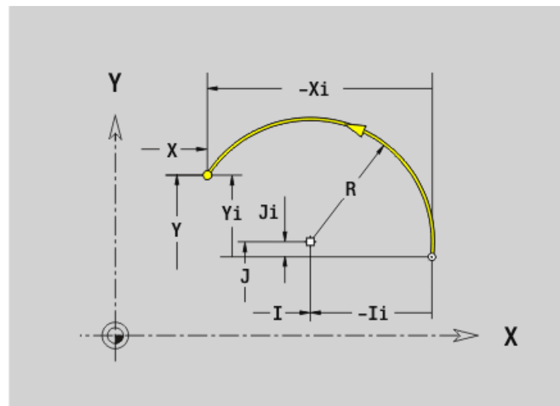


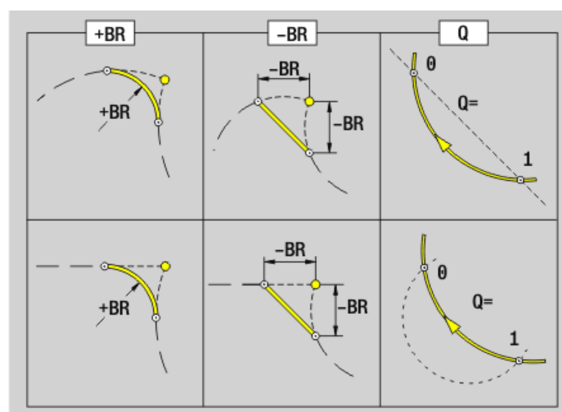


Circular arc in XY plane G172-Geo/G173-Geo

G172/G173 defines a circular arc in a contour of the XY plane. Direction of rotation: See help graphic







Parameters

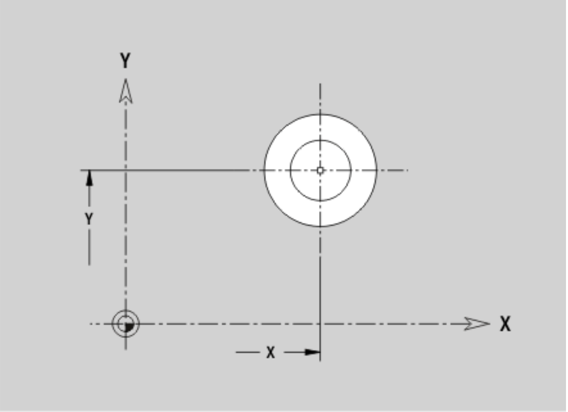
X	End point (radius)
Y	End point
R	Radius
I	Center in X direction (radius)
J	Center in Y direction
Q	Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):
	■ 0: Near point of intersection
	■ 1: Far point of intersection
BR	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
	■ No entry: Tangential transition
	■ BR=0: No tangential transition
	■ BR>0: Rounding radius
	■ BR<0: Width of chamfer

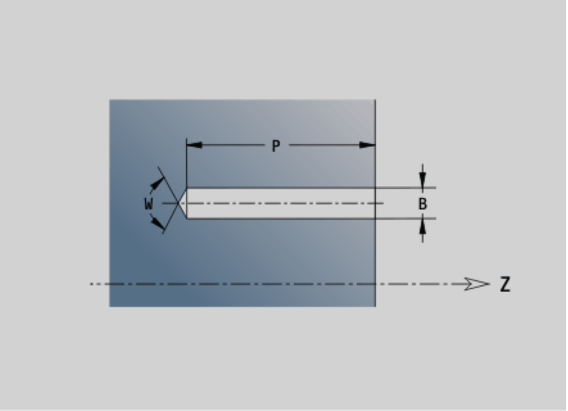
Programming

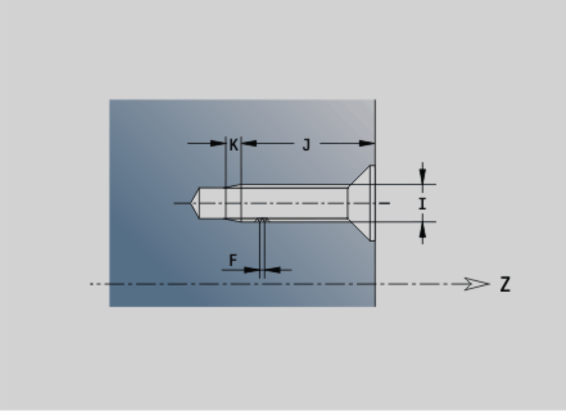
- **X, Y:** Absolute, incremental, modal or "?"
- **I, J:** Absolute or incremental
- End point must not be the starting point (no full circle).

Hole in XY plane G370-Geo

G370 defines a hole with countersinking and thread in the XY plane.







Parameters

X Center of hole (radius)
Y Center of hole
B Hole diameter
P Depth of hole (excluding point)
W Point angle (default: 180°)
R Sinking diameter
U Sinking depth
E Sinking angle
I Thread diameter
J Thread depth
K Start of thread (runout length)
F Thread pitch
V Left-hand or right-hand thread (default: 0)
- 0: Right-hand thread
- 1: Left-hand thread

A Angle to Z axis. Inclination of the hole
- Front face (range: –90° < A < 90°, default: 0°)
- Rear side (range: 90° < A < 270°, default: 180°)

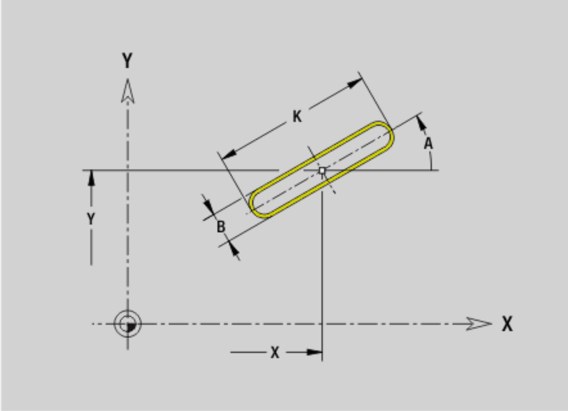
O Centering diameter

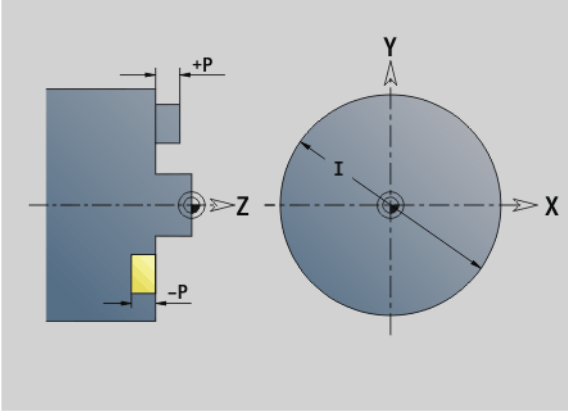
Linear slot in XY plane, G371-Geo

G371 defines the contour of a linear slot in the XY plane.

Parameters

- X Center of slot (radius)
- Y Center of slot
- K Slot length
- B Slot width
- A Position angle (reference: positive X axis; default: 0°)
- P Depth/height (default: "P" from G308)
  - P<0: Pocket
  - P>0: Island
- I Limit diameter (as cutting limit)
  - No entry: "X" from section code
  - "I" overwrites "X" from section code

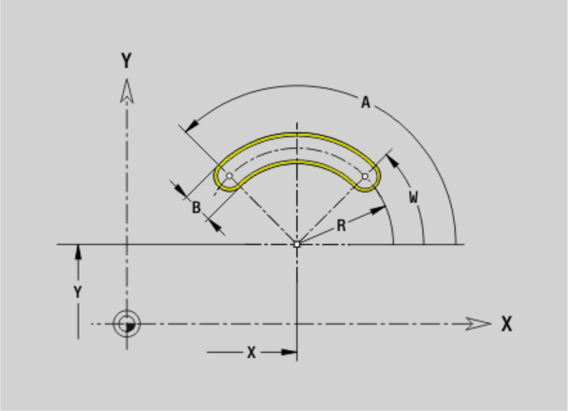


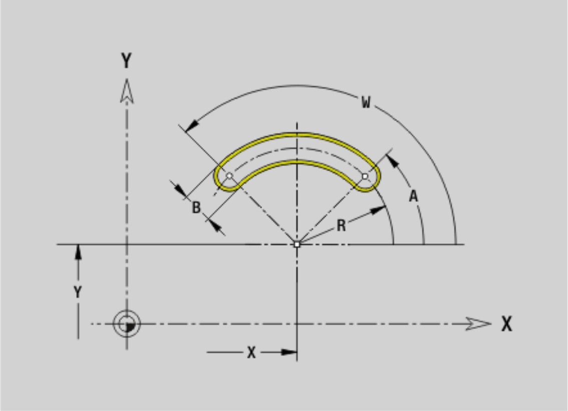


Circular slot in XY plane G372-Geo/G373-Geo

G372/G373 defines a circular slot in the XY plane.

- G372: Circular slot clockwise
- G373: Circular slot counterclockwise



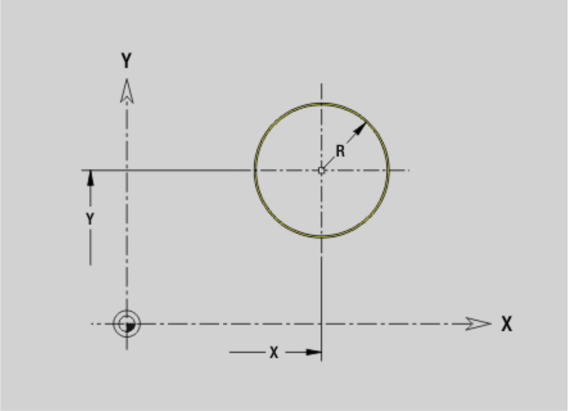


Parameters

X	Center of slot curvature (radius)
Y	Center of slot curvature
R	Curvature radius (reference: center point path of the slot)
A	Starting angle (reference: positive X axis; default: 0°)
W	Final angle (reference: positive X axis; default: 0°)
B	Slot width
P	Depth/height (default: "P" from G308) ■ P<0: Pocket ■ P>0: Island
I	Limit diameter (as cutting limit) ■ No entry: "X" from section code ■ "I" overwrites "X" from section code

Full circle in XY plane G374-Geo

G374 defines a full circle in the XY plane.

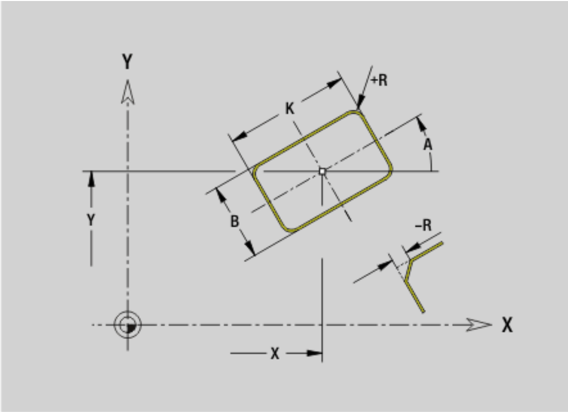


Parameters

X	Circle center (radius)
Y	Circle center
R	Circle radius
P	Depth/height (default: "P" from G308) ■ P<0: Pocket ■ P>0: Island
I	Limit diameter (as cutting limit) ■ No entry: "X" from section code ■ "I" overwrites "X" from section code

Rectangle in XY plane G375-Geo

G375 defines a rectangle in the XY plane.

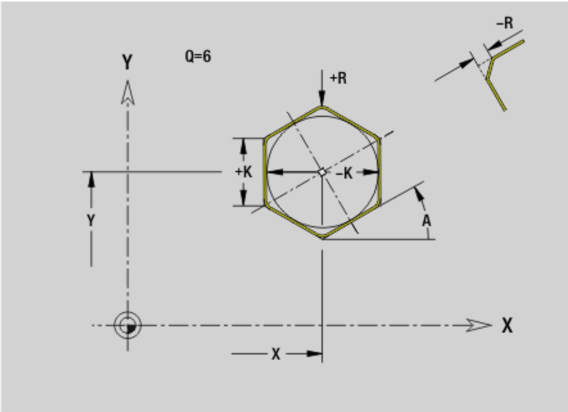


Parameters

- X Center of rectangle (radius)
- Y Center of rectangle
- A Position angle (reference: positive X axis; default: 0°)
- K Length of rectangle
- B Width of rectangle
- R Chamfer/rounding (default: 0)
  - R>0: Radius of rounding arc
  - R<0: Chamfer width
- P Depth/height (default: "P" from G308)
  - P<0: Pocket
  - P>0: Island
- I Limit diameter (as cutting limit)
  - No entry: "X" from section code
  - "I" overwrites "X" from section code

Eccentric polygon in XY plane G377-Geo

G377 defines the contour of an eccentric polygon in the XY plane.



Parameters

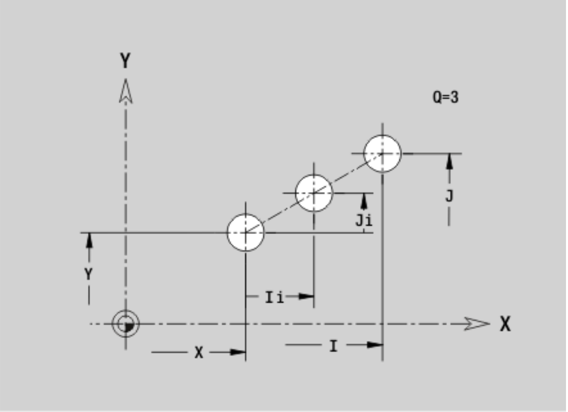
- X Center point of polygon (radius)
- Y Center point of polygon
- Q Number of corners (Q >= 3)
- A Position angle (reference: positive X axis; default: 0°)
- K Edge length / width across flats
  - K>0: Edge length
  - K<0: Width across flats (inside diameter)
- R Chamfer/rounding—default: 0
  - R>0: Radius of rounding arc
  - R<0: Chamfer width
- P Depth/height (default: "P" from G308)
  - P<0: Pocket
  - P>0: Island
- I Limit diameter (as cutting limit)
  - No entry: "X" from section code
  - "I" overwrites "X" from section code

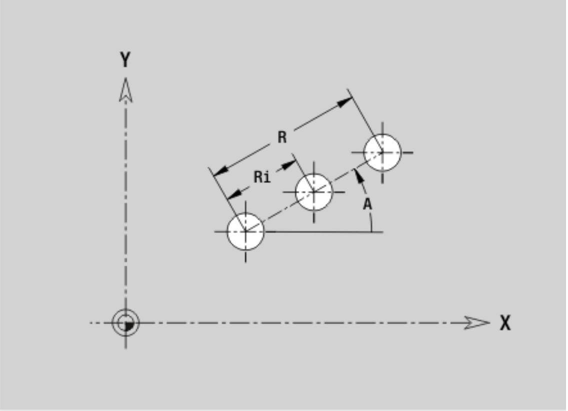
Linear pattern in XY plane, G471-Geo

G471 defines a linear pattern in the XY plane. G471 affects the hole or figure defined in the following block (G370 to G375, G377).

Parameters

Q	Number of figures
X	1st point of pattern (radius)
Y	1st point of pattern
I	End point of pattern (X direction; radius)
J	End point of pattern (Y direction)
Ii	Distance in X direction between two figures
Ji	Distance in Y direction between two figures
A	Position angle of longitudinal axis of pattern (reference: positive X axis)
R	Length (overall length of pattern)
Ri	Pattern distance (distance between two figures)





Programming notes

- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

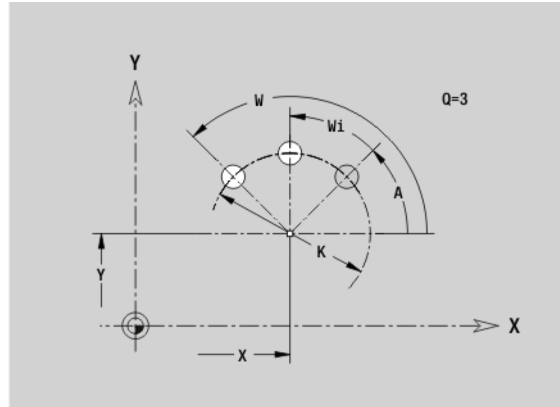
Circular pattern in XY plane, G472 Geo

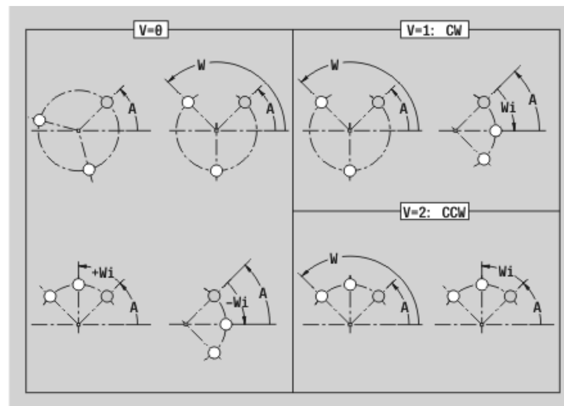
G472 defines a circular pattern in the XY plane. G472 is effective for the figure defined in the following block (G370 to G375, G377).

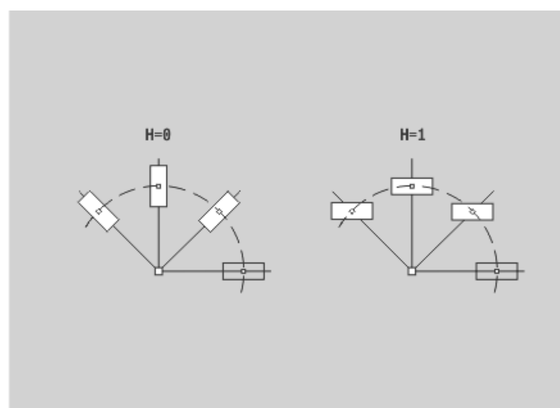
Parameters

- Q Number of figures
- K Diameter (pattern diameter)
- A Starting angle—position of the first figure (reference: positive X axis; default: 0°)
- W Final angle—position of the last figure (reference: positive X axis; default: 360°)
- Wi Angle between two figures
- V Direction—orientation (default: 0)
  - V=0, without W: Figures are arranged on a full circle
  - V=0, with W: Figures are arranged on the longer circular arc
  - V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
  - V=1, with W: Clockwise
  - V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
  - V=2, with W: Counterclockwise
  - V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)
- X Center of pattern (radius)
- Y Center of pattern
- H Position of the figures (default: 0)
  - 0: Normal position—the figures are rotated about the circle center (rotation)
  - 1: Original position—the position of the figures relative to the coordinate system remains unchanged (translation)

> - Program the hole/figure in the following block without a center. Exception: **circular slot**.
> - The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

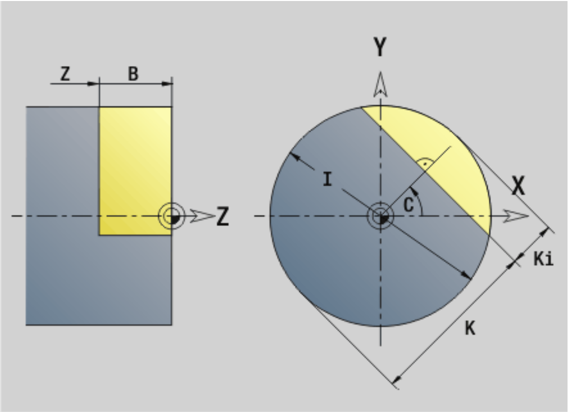






Single surface in XY plane G376-Geo

G376 defines a surface in the XY plane.



Parameters
Z	Reference edge (default: "Z" from section code)
K	Residual depth
Ki	Depth
B	Width (reference: reference edge Z)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction

I	Limit diameter (as cutting limit and as reference for K/Ki)

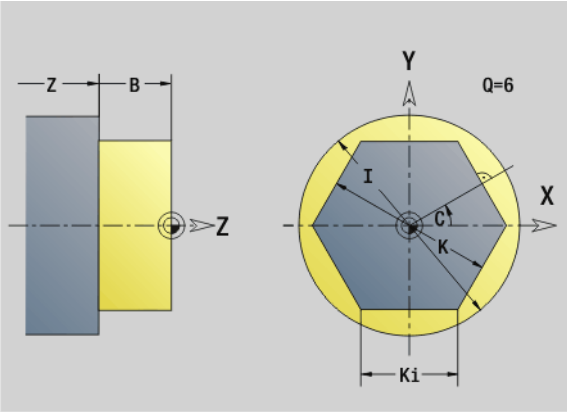
- No entry: "X" from section code
- "I" overwrites "X" from section code

C	Spindle angle of surface normal (default: "C" from section code)

> Whether the surface lies on the front face or rear side has no effect on the evaluation of the algebraic sign for "width B."

Centric polygon in XY plane G477-Geo

G477 defines polygonal surfaces in the XY plane.



Parameters
Z	Reference edge (default: "Z" from section code)
K	Width across flats (inscribed circle diameter)
Ki	Length of side
B	Width (reference: reference edge Z)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction

C	Spindle angle of surface normal (default: "C" from section code)
Q	Number of sides (Q >= 2)
I	Limit diameter (as cutting limit)

- No entry: "X" from section code
- "I" overwrites "X" from section code

> Whether the surface lies on the front face or rear side has no effect on the evaluation of the algebraic sign for "width B."

6.3 Contours in the YZ plane

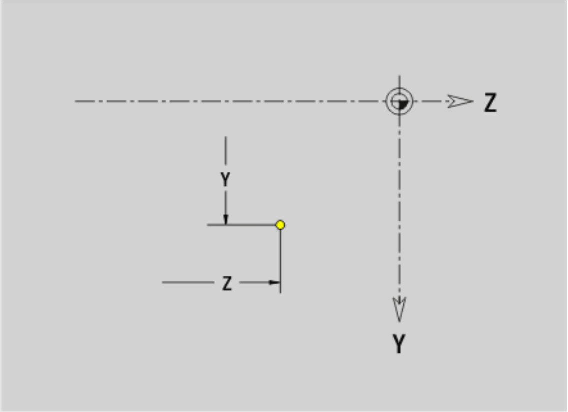
Starting point of contour in YZ plane G180-Geo

G180 defines the starting point of a contour in the YZ plane.

Parameters

Y Starting point of contour

Z Starting point of contour



Line segment in YZ plane G181-Geo

G181 defines a line segment in a contour of the YZ plane.

Parameters

Y End point

Z End point

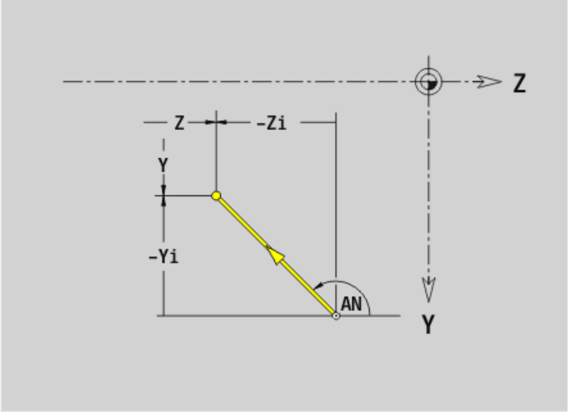
AN Angle to positive Z axis

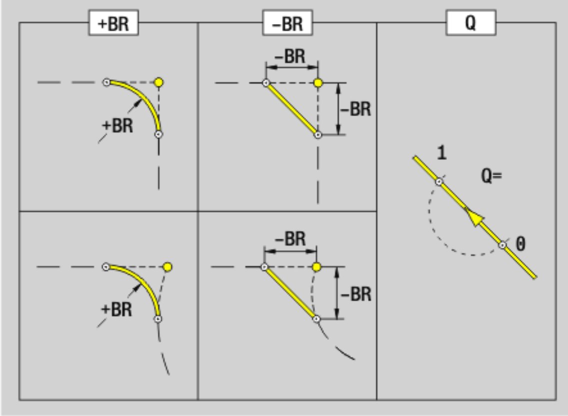
Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

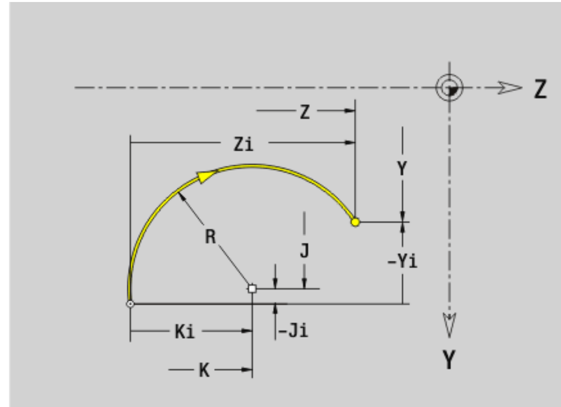


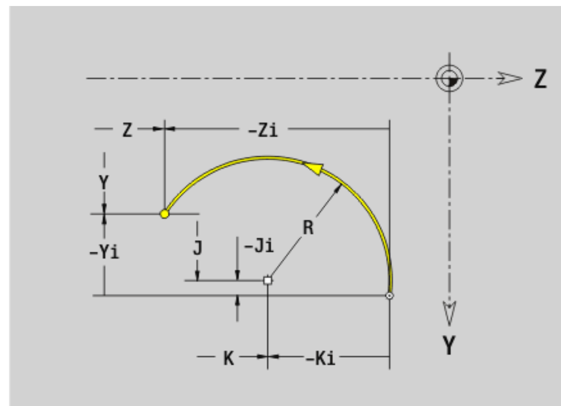


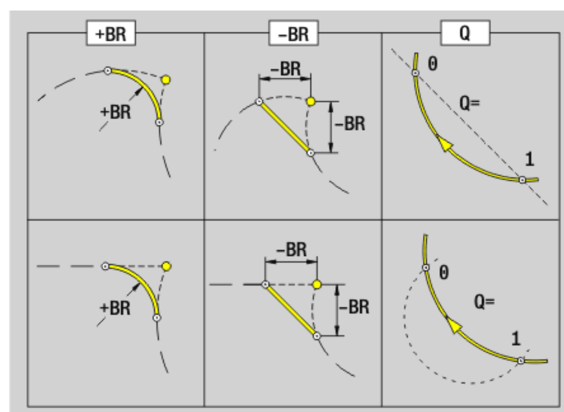
Programming Y, Z: Absolute, incremental, modal or "?"

Circular arc in YZ plane G182-Geo/G183-Geo

G182/G183 defines a circular arc in a contour of the YZ plane. Direction of rotation: See help graphic







Parameters

Y End point (radius)
Z End point
J Center (Y direction)
K Center (Z direction)
R Radius
Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

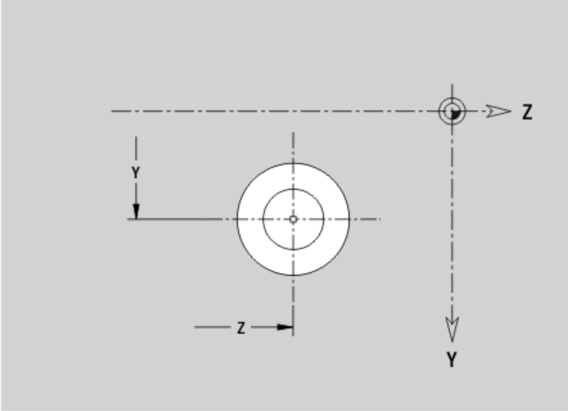
- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

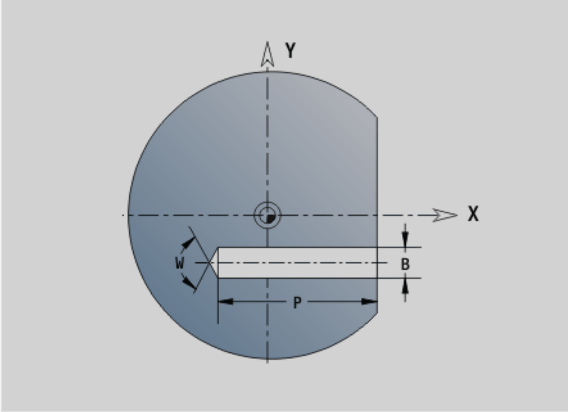
Programming

- **Y, Z:** Absolute, incremental, modal or "?"
- **J, K:** Absolute or incremental
- End point must not be the starting point (no full circle).

Hole in YZ plane G380-Geo

G380 defines a single hole with countersinking and thread in the YZ plane.



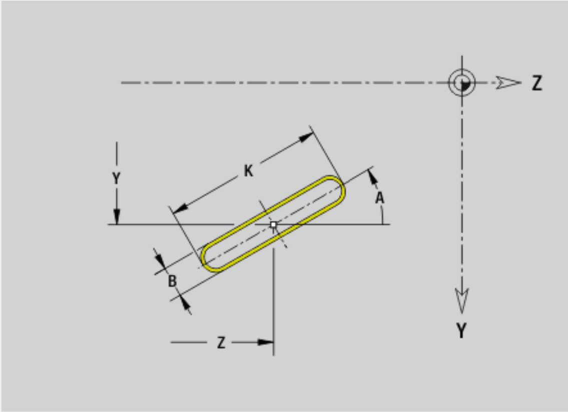


Parameters

- Y Center of hole
- Z Center of hole
- B Hole diameter
- P Depth of hole (excluding point)
- W Point angle (default: 180°)
- R Sinking diameter
- U Sinking depth
- E Sinking angle
- I Thread diameter
- J Thread depth
- K Start of thread (runout length)
- F Thread pitch
- V Left-hand or right-hand thread (default: 0)
  - 0: Right-hand thread
  - 1: Left-hand thread
- A Angle to X axis (range: –90° < A < 90°)
- O Centering diameter

Linear slot in YZ plane, G381-Geo

G381 defines the contour of a linear slot in the YZ plane.



Parameters

- Y Center of slot
- Z Center of slot
- X Reference diameter
  - No entry: "X" from section code
  - "X" overwrites "X" from section code
- A Position angle (reference: positive Z axis; default: 0°)
- K Slot length
- B Slot width
- P Pocket depth (default: "P" from G308)

Circular slot in YZ plane G382-Geo/G383-Geo

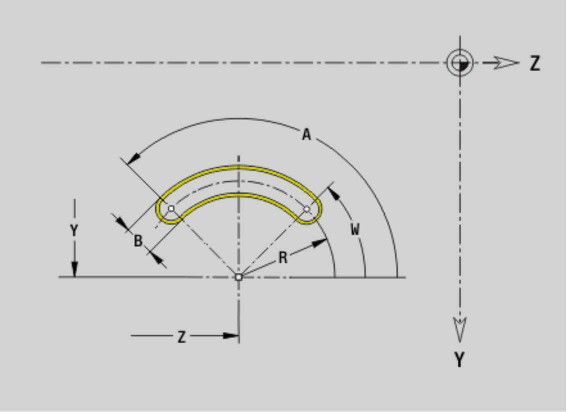
G382/G383 defines a circular slot in the YZ plane.

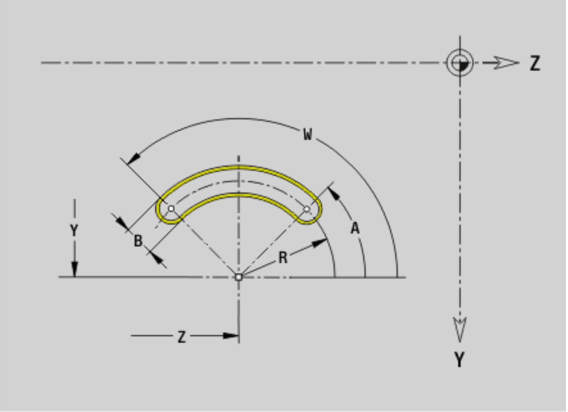
- G382: Circular slot clockwise
- G383: Circular slot counterclockwise

Parameters

Y Center of slot curvature
Z Center of slot curvature
X Reference diameter
- No entry: "X" from section code
- "X" overwrites "X" from section code

R Radius (reference: center point path of the slot)
A Starting angle (reference: X axis; default: 0°)
W Final angle (reference: X axis; default: 0°)
B Slot width
P Pocket depth (default: "P" from G308)





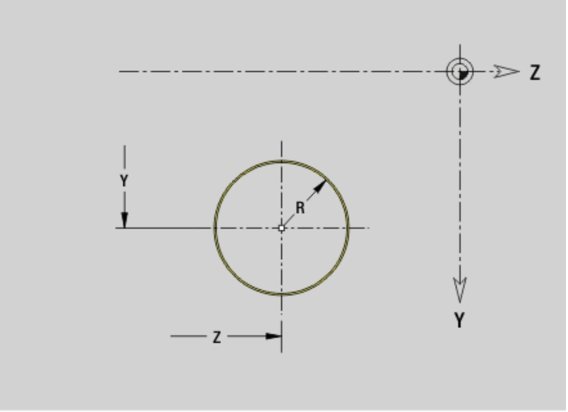
Full circle in YZ plane G384-Geo

G384 defines a full circle in the YZ plane.

Parameters

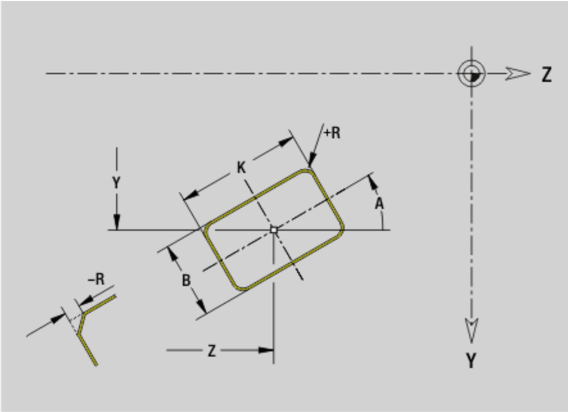
Y Center of circle
Z Center of circle
X Reference diameter
- No entry: "X" from section code
- "X" overwrites "X" from section code

R Circle radius
P Pocket depth (default: "P" from G308)



Rectangle in YZ plane G385-Geo

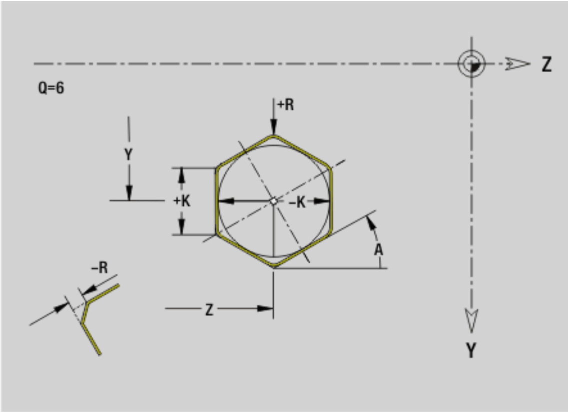
G385 defines a rectangle in the YZ plane.



Parameters
- Y Center of rectangle
- Z Center of rectangle
- X Reference diameter
  - No entry: "X" from section code
  - "X" overwrites "X" from section code
- A Position angle (reference: positive Z axis; default: 0°)
- K Length of rectangle
- B Width of rectangle
- R Chamfer/rounding (default: 0)
  - R>0: Radius of rounding arc
  - R<0: Chamfer width
- P Pocket depth (default: "P" from G308)

Eccentric polygon in YZ plane G387-Geo

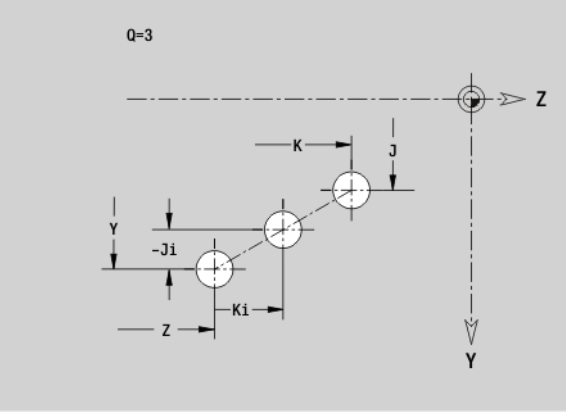
G387 defines the contour of an eccentric polygon in the YZ plane.



Parameters
- Y Center point of polygon
- Z Center point of polygon
- X Reference diameter
  - No entry: "X" from section code
  - "X" overwrites "X" from section code
- Q Number of corners (Q >= 3)
- A Position angle (reference: positive Z axis; default: 0°)
- K Edge length / width across flats
  - K>0: Edge length
  - K<0: Width across flats (inside diameter)
- R Chamfer/rounding—default: 0
  - R>0: Radius of rounding arc
  - R<0: Chamfer width
- P Pocket depth (default: "P" from G308)

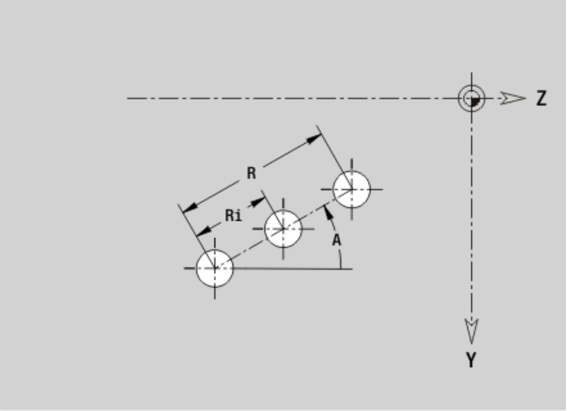
Linear pattern in YZ plane, G481-Geo

G481 defines a linear pattern in the YZ plane. G481 is effective for the figure defined in the following block (G380 to G385, G387).



Parameters

Q	Number of figures
Y	1st point of pattern
Z	1st point of pattern
J	End point of pattern (Y direction)
K	End point of pattern (Z direction)
Ji	Distance between two figures (in Y direction)
Ki	Distance between two figures (in Z direction)
A	Position angle of longitudinal axis of pattern (reference: positive Z axis)
R	Length (overall length of pattern)
Ri	Pattern distance (distance between two figures)



Programming notes

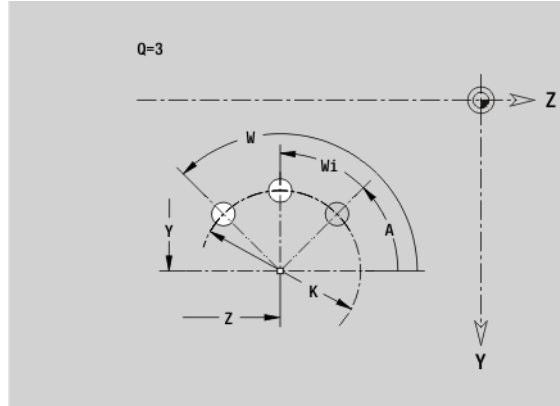
- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

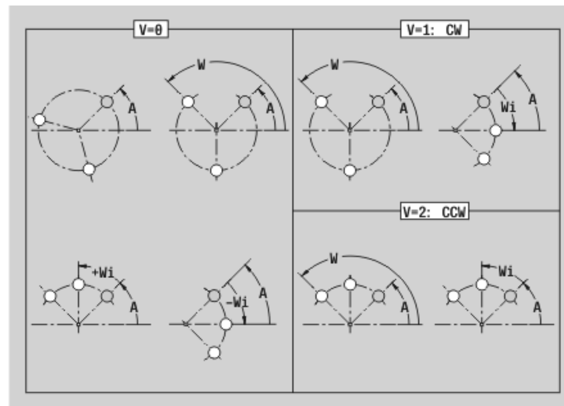
Circular pattern in YZ plane, G482-Geo

G482 defines a circular pattern in the YZ plane. G482 is effective for the figure defined in the following block (G380 to G385, G387).

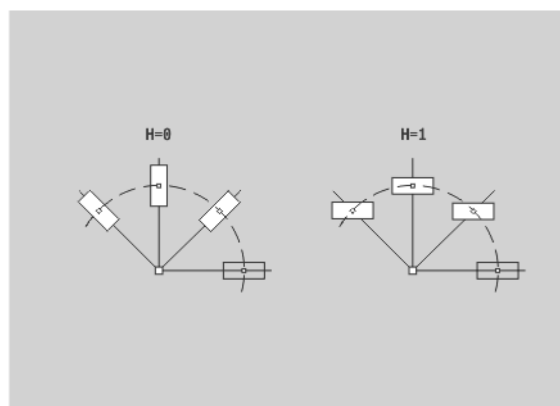
Parameters

- Q — Number of figures
- K — Diameter (pattern diameter)
- A — Starting angle—position of the first figure; reference: Z axis (default: 0°)
- W — End angle—position of the last figure; reference: Z axis (default: 360°)
- Wi — Angle between two figures
- V — Direction—orientation (default: 0)
  - V=0, without W: Figures are arranged on a full circle
  - V=0, with W: Figures are arranged on the longer circular arc
  - V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
  - V=1, with W: Clockwise
  - V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
  - V=2, with W: Counterclockwise
  - V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)
- Y — Center of pattern
- Z — Center of pattern
- H — Position of the figures (default: 0)
  - 0: Normal position—the figures are rotated about the circle center (rotation)
  - 1: Original position—the position of the figures relative to the coordinate system remains unchanged (translation)



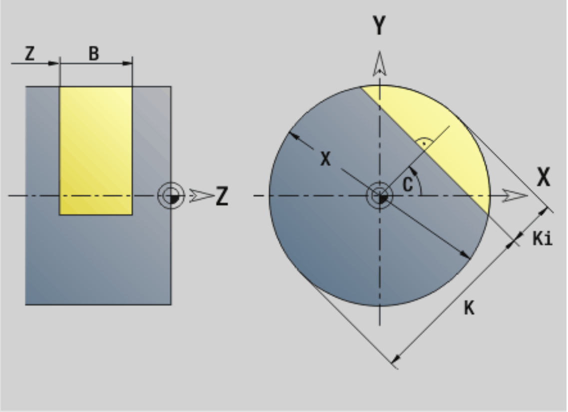


- Program the hole/figure in the following block without a center. **Exception: circular slot**.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.



Single surface in YZ plane G386-Geo

G386 defines a surface in the YZ plane.



Parameters

Z Reference edge
K Residual depth
Ki Depth
B Width (reference: reference edge Z)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction

X Reference diameter

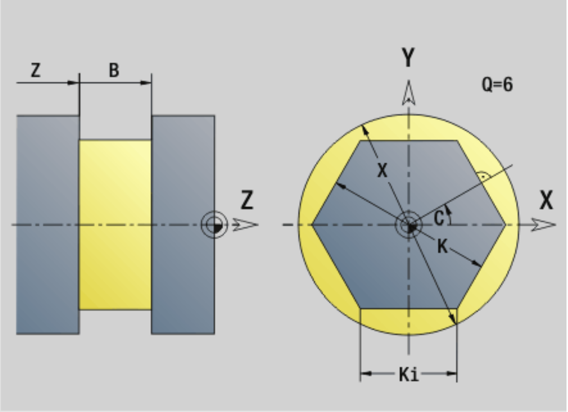
- No entry: "X" from section code
- "X" overwrites "X" from section code

C Spindle angle of surface normal (default: "C" from section code)

The **reference diameter X** limits the surface to be machined.

Centric polygon in YZ plane G487-Geo

G487 defines polygonal surfaces in the YZ plane.



Parameters

Z Reference edge
K Width across flats (inscribed circle diameter)
Ki Length of side
B Width (reference: reference edge Z)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction

X Reference diameter

- No entry: "X" from section code
- "X" overwrites "X" from section code

C Spindle angle of surface normal (default: "C" from section code)
Q Number of sides (Q >= 2)

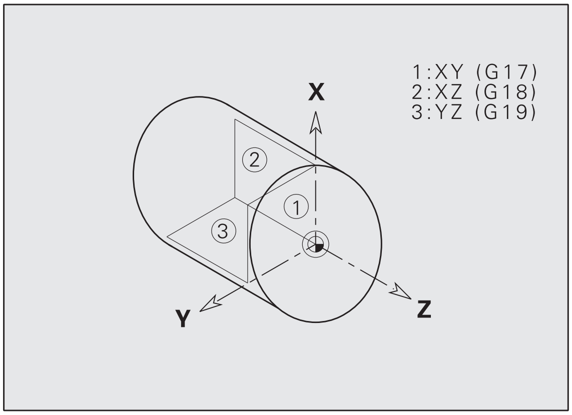
The **reference diameter X** limits the surface to be machined.

6.4 Working planes

Y-axis machining

When programming drilling or milling operations with the Y axis, you need to define the working plane.

If no working plane is programmed, the Control assumes a turning operation or a milling operation with the C axis (G18 XZ plane).



G17 XY plane (front or rear face)

Milling cycles are executed in the XY plane, with the depth feed for milling and drilling cycles in the Z direction.

G18 XZ plane (turning)

In the XZ plane, "normal turning operations" as well as drilling and milling operations are executed with the C axis.

G19 YZ plane (lateral view / lateral surface)

Milling cycles are executed in the YZ plane, with the depth feed for milling and drilling cycles in the X direction.

Tilting the working plane G16

G16 executes the following transformations and rotations:

- Shifts the coordinate system to the position I, K
- Rotates the coordinate system by the angle B; reference point: I, K
- Shifts, if programmed, the coordinate system by U and W in the rotated coordinate system

Parameters

B	Plane angle; reference: positive Z axis
I	Plane reference in X direction (radius)
K	Plane reference in Z direction
U	Shift in X direction
W	Shift in Z direction
Q	Enable/disable tilting the working plane

- 0: Disable tilted working plane function
- 1: Tilt working plane
- 2: Restore previous G16 plane

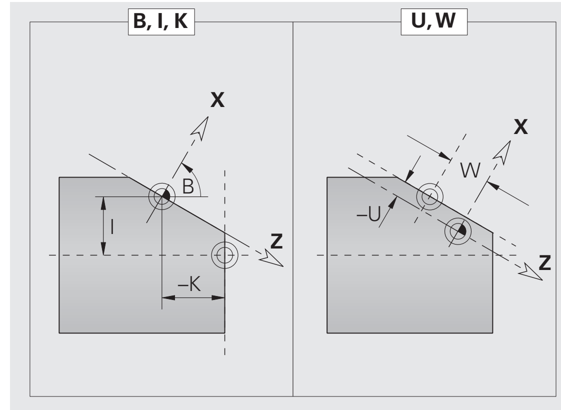
G16 Q0 resets the working plane. The zero point and coordinate system defined before G16 are then in effect again.

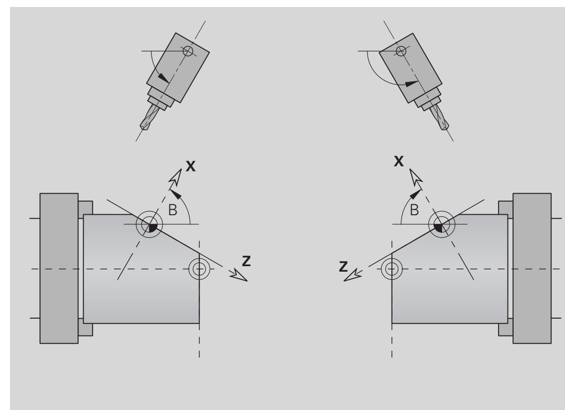
G16 Q2 restores the previous G16 plane.

The positive Z axis is the reference axis for the "plane angle B." This also applies to a mirrored coordinate system.

Please note:

- X is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
- Mirroring the coordinate system has no effect on the reference axis of the tilt angle ("B axis angle" of the tool call).
- Other zero point shifts are not permitted while G16 is active.





Example: "G16"

```
. . .
MACHINING
...
N.. G19
N.. G15 B130
N.. G16 B130 I59 K0 Q1
N.. G1 x.. Z.. Y..
N.. G16 Q0
. . .
```

6.5 Tool positioning in the Y axis

Rapid traverse G0

G0 moves the tool at rapid traverse along the shortest path to the "target point X, Y, Z."

Parameters
X Diameter—target point
Z Length—target point
Y Length—target point

Programming X, Y, Z: Absolute, incremental or modal

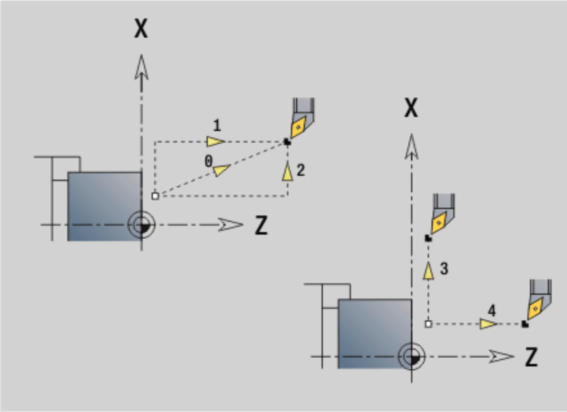
Approach tool change point G14

G14 moves at rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change position.

Parameter
Q Sequence (default: 0)

- 0: Move simultaneously in X and Z axes (diagonal path)
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: Only X direction, Z remains unchanged
- 4: Only Z direction, X remains unchanged
- 5: Y direction only
- 6: Move simultaneously in X, Y and Z axes (diagonal path)

If Q=0 to 4, the Y axis does not move.



Rapid traverse to machine coordinates G701

G701 moves the tool at rapid traverse along the shortest path to the "target point X, Y, Z."

Parameters

X	End point (diameter)
Y	End point
Z	End point

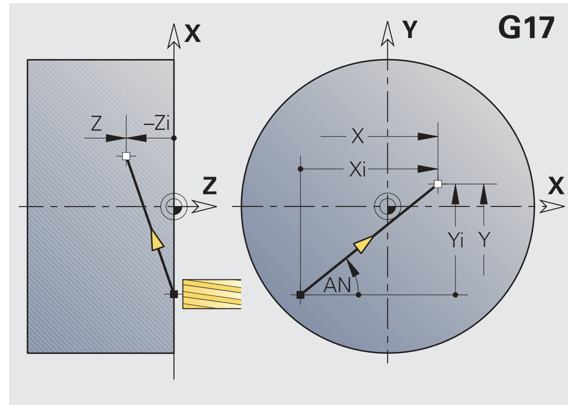
"X, Y, Z" refer to the **machine zero point** and the **slide reference point**.

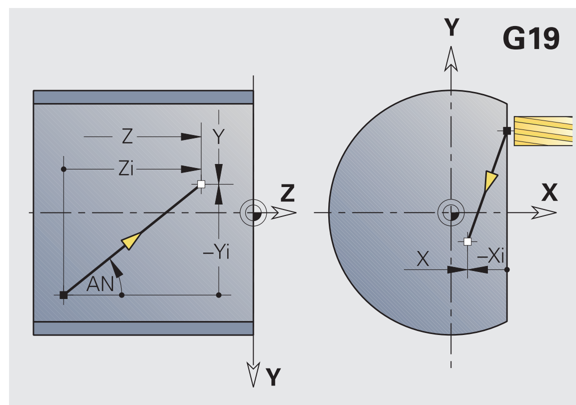
6.6 Linear and circular movements in the Y axis

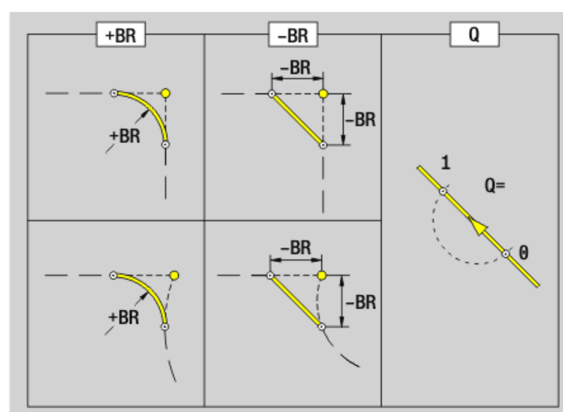
Milling: Linear movement G1

G1 moves the tool on a linear path at the feed rate to the "end point." The execution of G1 varies depending on the **working plane**:

- G17 Interpolation in the XY plane
  - Infeed in Z direction
  - Angle A—reference: positive X axis
- G18 Interpolation in the XZ plane
  - Infeed in Y direction
  - Angle A—reference: negative Z axis
- G19 Interpolation in the YZ plane
  - Infeed in X direction
  - Angle A—reference: positive Z axis







Parameters

X	End point (diameter)
Y	End point
Z	End point
AN	Angle (reference: depends on the working plane)
Q	Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- 0: Near point of intersection
- 1: Far point of intersection

BR Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- BR=0: No tangential transition
- BR>0: Rounding radius
- BR<0: Width of chamfer

BE Special feed factor for chamfer/rounding arc (default: 1)

Special feed rate = active feed rate * BE (0 < BE <= 1)

Programming X, Y, Z: Absolute, incremental or modal or "?"

Milling: Circular movement G2, G3—incremental center coordinates

G2/G3 moves the tool in a circular arc at the feed rate to the "end point."

The execution of G2/G3 varies depending on the **working plane**:

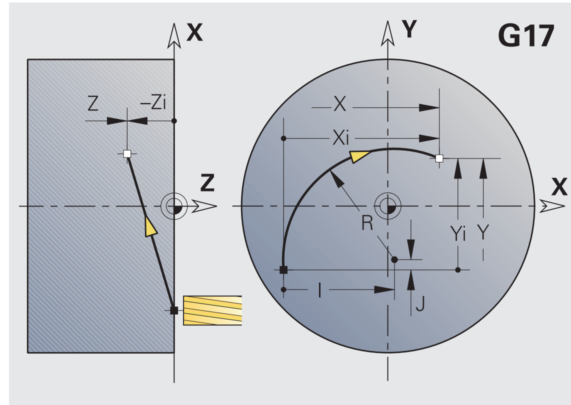
- G17 Interpolation in the XY plane
  - Infeed in Z direction
  - Center definition: with I, J
- G18 Interpolation in the XZ plane
  - Infeed in Y direction
  - Center definition: with I, K
- G19 Interpolation in the YZ plane
  - Infeed in X direction
  - Center definition: with J, K

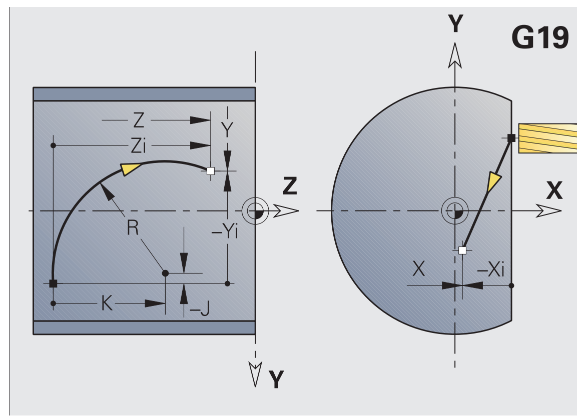
Parameters

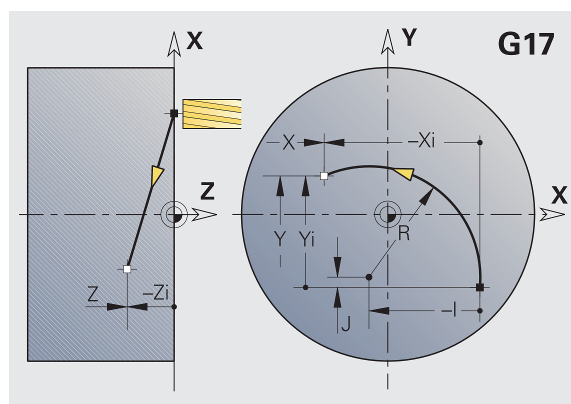
X	End point (diameter)
Y	End point
Z	End point
I	Incremental center point (radius)
J	Incremental center point
K	Incremental center point
R	Radius
Q	Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0): ■ 0: Near point of intersection ■ 1: Far point of intersection
BR	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point. ■ No entry: Tangential transition ■ BR=0: No tangential transition ■ BR>0: Rounding radius ■ BR<0: Width of chamfer
BE	Special feed factor for chamfer/rounding arc (default: 1) Special feed rate = active feed rate * BE (0 < BE <= 1)

If you do not program the center, the Control automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.

Programming X, Y, Z: Absolute, incremental or modal or "?"





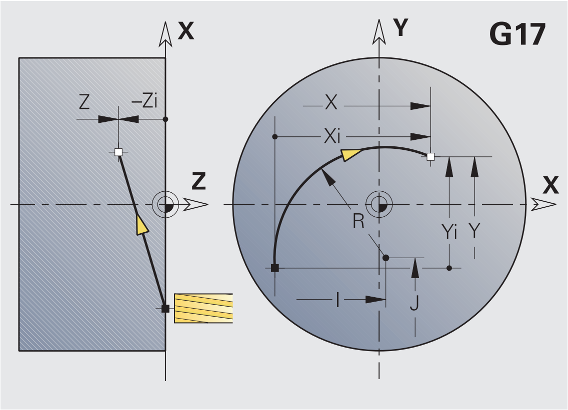


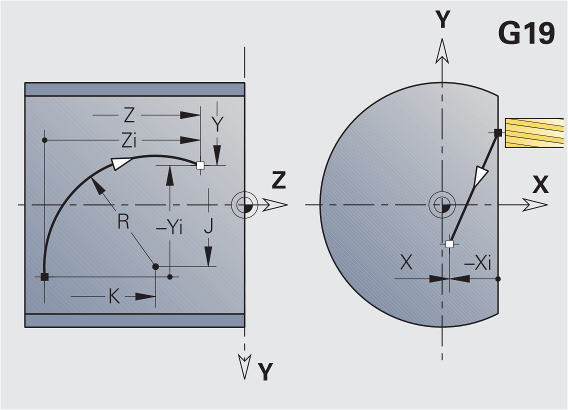
Milling: Circular movement G12, G13—absolute center coordinates

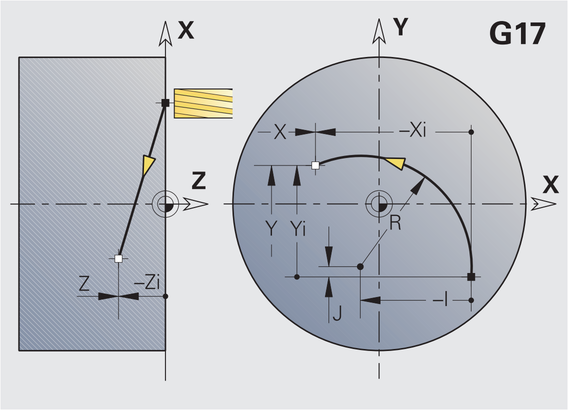
G12/G13 moves the tool in a circular arc at the feed rate to the "end point."

The execution of G12/G13 varies depending on the **working plane**:

- G17 Interpolation in the XY plane
  - Infeed in Z direction
  - Center definition: with I, J
- G18 Interpolation in the XZ plane
  - Infeed in Y direction
  - Center definition: with I, K
- G19 Interpolation in the YZ plane
  - Infeed in X direction
  - Center definition: with J, K







Parameters

X	End point (diameter)
Y	End point
Z	End point
I	Absolute center point (radius)
J	Absolute center point
K	Absolute center point
R	Radius
Q	Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

B	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E	Special feed factor for the chamfer/rounding (default: 1)

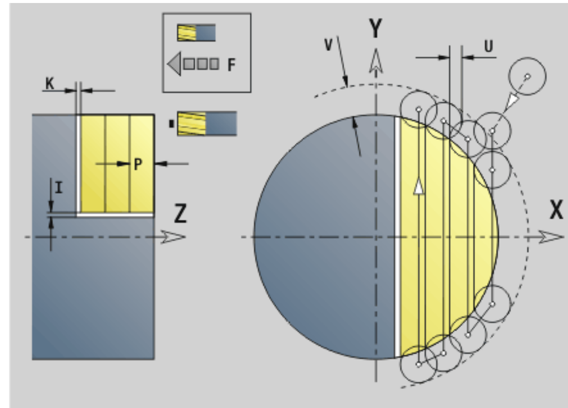
Special feed rate = active feed rate * E (0 < E <= 1)

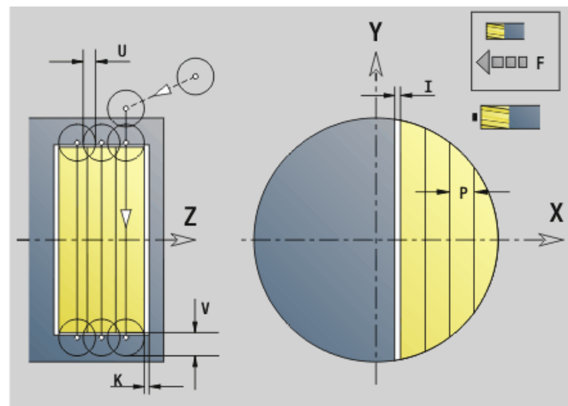
If you do not program the center, the Control automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.

Programming X, Y, Z: Absolute, incremental or modal or "?"

6.7 Milling cycles for the Y axis

Area milling—roughing G841





G841 roughs surfaces defined with G376-Geo (XY plane) or with G386-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters

ID	Milling contour—name of the contour to be milled
NS	Block number—reference to the contour description
P	Milling depth (maximum infeed in the working plane)
I	Oversize in X direction
K	Oversize in Z direction
U	(Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5). Overlap = U*milling diameter
V	Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5). Overrun = V*milling diameter
F	Feed rate for infeed (default: active feed rate)
RB	Retraction plane (default: back to starting position) ■ XY plane: Retraction position in Z direction ■ YZ plane: Retraction position in X direction (diameter)

Oversizes are taken into account:

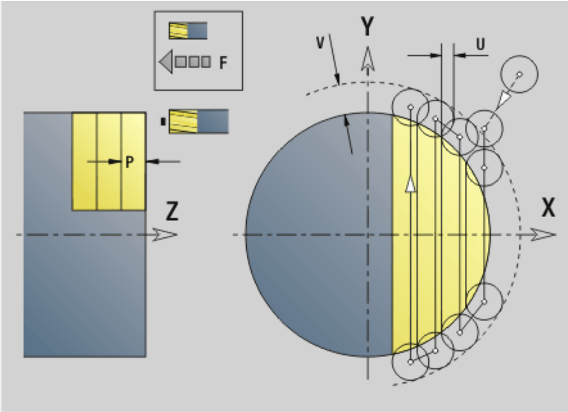
- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

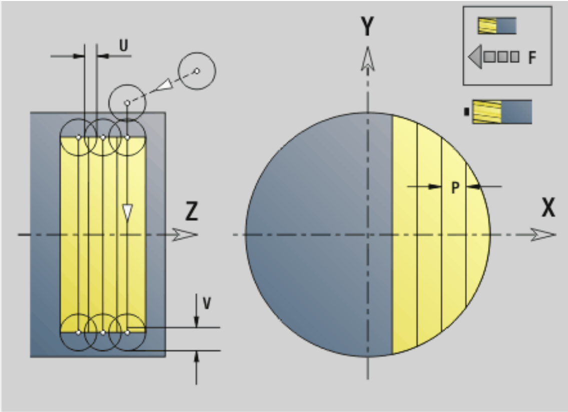
Cycle run

1. Starting position (X, Y, Z, C) is the position before the cycle begins.
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths).
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to retraction plane RB.

Area milling—finishing G842

G842 finishes surfaces defined with G376-Geo (XY plane) or G386-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.





Parameters

ID Milling contour—name of the contour to be milled

NS Block number—reference to the contour description

P Milling depth (maximum infeed in the working plane)

H Cutting direction for side finishing (default: 0)

- H=0: Up-cut milling
- H=1: Climb milling

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5).

Overrun = V*milling diameter

F Feed rate for infeed (default: active feed rate)

RB Retraction plane (default: back to starting position)

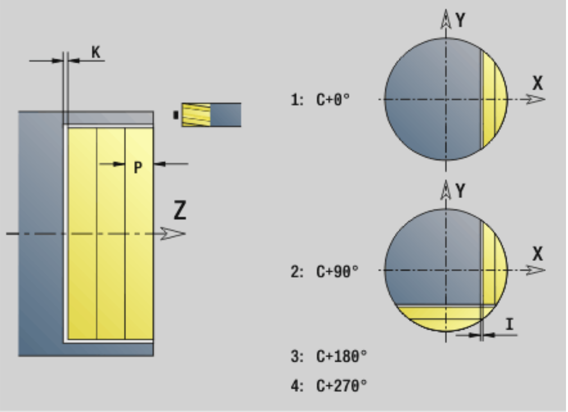
- XY plane: Retraction position in Z direction
- YZ plane: Retraction position in X direction (diameter)

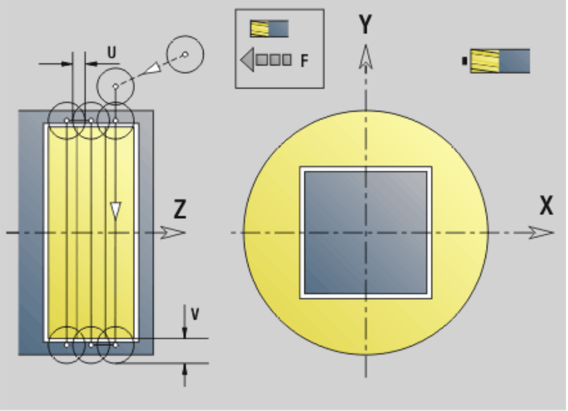
Cycle run

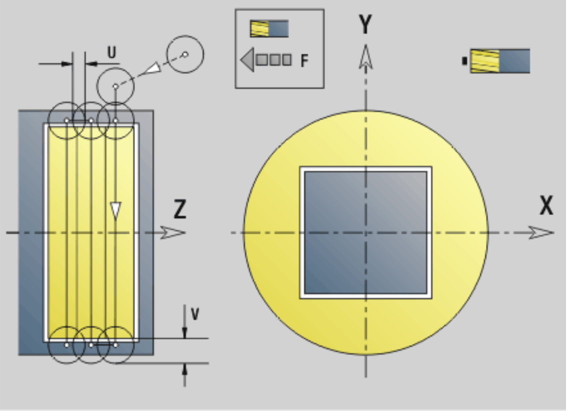
1. Starting position (X, Y, Z, C) is the position before the cycle begins.
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths).
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to retraction plane RB.

Centric polygon milling—roughing G843

G843 roughs centric polygons defined with G477-Geo (XY plane) or G487-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.







Parameters

ID	Milling contour—name of the contour to be milled
NS	Block number—reference to the contour description
P	Milling depth (maximum infeed in the working plane)
I	Oversize in X direction
K	Oversize in Z direction
U	(Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5). Overlap = U*milling diameter
V	Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5). Overrun = V*milling diameter
F	Feed rate for infeed (default: active feed rate)
RB	Retraction plane (default: back to starting position) ■ XY plane: Retraction position in Z direction ■ YZ plane: Retraction position in X direction (diameter)

Oversizes are taken into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

Cycle run

1. Starting position (X, Y, Z, C) is the position before the cycle begins.
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the spindle positions.
3. Spindle turns to the first position. The tool moves to the safety clearance and plunges to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. The tool returns to "retraction plane J." The spindle turns to the next position. The tool moves to the safety clearance and plunges to the first milling depth.
8. Repeat steps 4 to 7 until all polygonal surfaces are milled.
9. Returns to retraction plane RB.

Centric polygon milling—finishing G844

G844 finishes centric polygons defined with G477-Geo (XY plane) or with G487-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters

ID Milling contour—name of the contour to be milled

NS Block number—reference to the contour description

P Milling depth (maximum infeed in the working plane)

H Cutting direction for side finishing (default: 0)

- H=0: Up-cut milling
- H=1: Climb milling

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

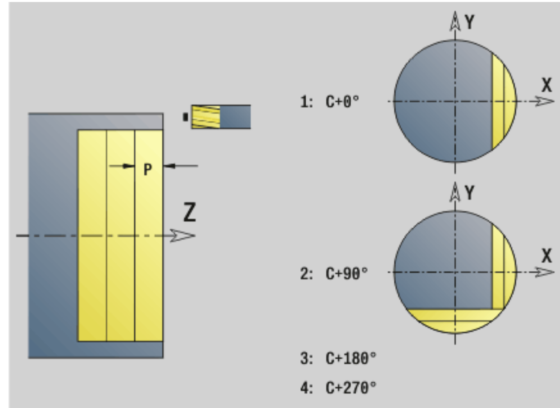
V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5).

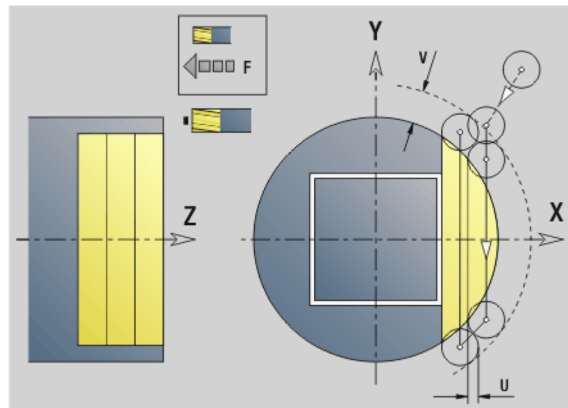
Overrun = V*milling diameter

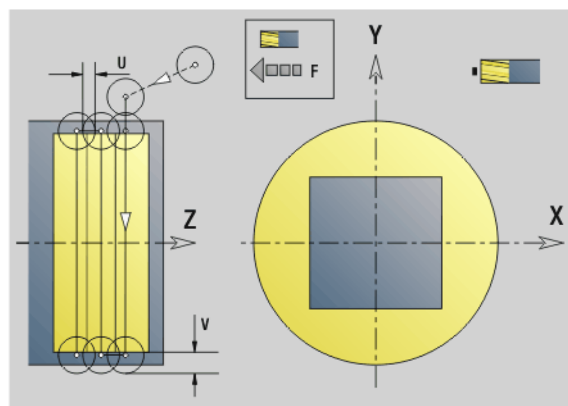
F Feed rate for infeed (default: active feed rate)

RB Retraction plane (default: back to starting position)

- XY plane: Retraction position in Z direction
- YZ plane: Retraction position in X direction (diameter)







Cycle run

1. Starting position (X, Y, Z, C) is the position before the cycle begins.
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the spindle positions.
3. Spindle turns to the first position. The tool moves to the safety clearance and plunges to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. The tool returns to "retraction plane J." The spindle turns to the next position. The tool moves to the safety clearance and plunges to the first milling depth.
8. Repeat steps 4 to 7 until all polygonal surfaces are milled.
9. Returns to retraction plane RB.

Pocket milling—roughing G845 (Y axis)

G845 roughs closed contours that are defined in the XY or YZ plane in the program sections:

- FACE_Y
- REAR SIDE_Y
- LATERAL_Y

Choose one of the following **plunge strategies**, depending on the milling cutter you are using:

- Plunge vertically
- Plunge at a pre-drilled position
- Plunge in a reciprocating or helical motion

When "plunging at a pre-drilled position," you have the following alternatives:

- **Calculate positions, drill, mill.** The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with "G845 A1 ..."
  - Drill holes with "G71 NF ..."
  - Call cycle "G845 A0 ..." The cycle positions the tool above the hole; the tool plunges and mills the pocket.
- **Drill, mill.** The machining process is performed in the following steps:
  - Drill a hole inside the pocket with "G71 ..."
  - Position the milling cutter above the hole and call "G845 A0 ..." The tool plunges and mills the section.

If the pocket consists of multiple sections, G845 takes all the sections of the pocket into account for drilling and milling. Call "G845 A0 .." separately for each section when calculating the hole positions without "G845 A1 ..".

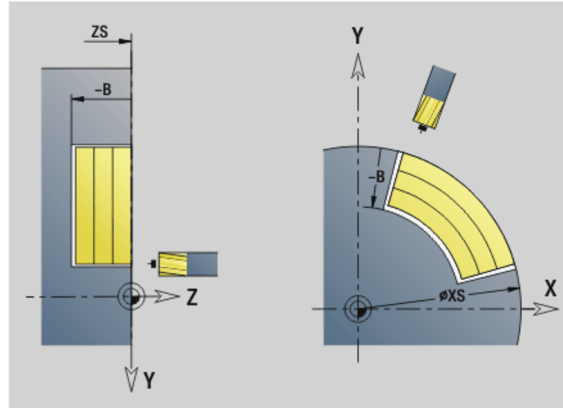
G845 takes the following oversizes into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

Program oversizes for calculating the hole positions **and** for milling.

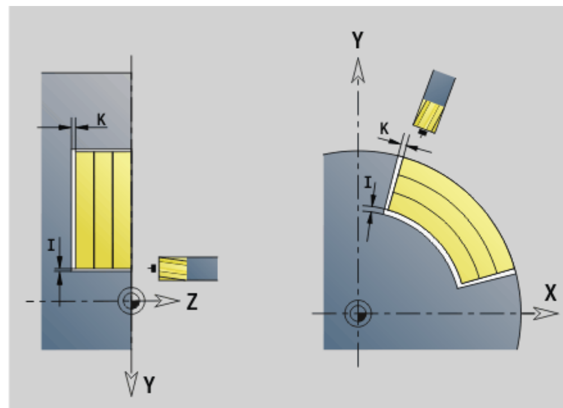
G845 (Y axis)—Calculating hole positions

"G845 A1 .." calculates the hole positions and stores them at the reference specified in "NF." The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling "G845 A1 ..". Program only the parameters given in the following table.



See also:

- G845—Fundamentals: Page 508
- G845—Milling: Page 510



Parameters—Calculating hole positions

ID Milling contour—name of the contour to be milled

NS Starting block no. of contour

- Figures: Block number of the figure
- Free closed contour: A contour element (not starting point)

B Milling depth (default: depth from the contour description)

XS Milling top edge—lateral surface (replaces the reference plane from the contour definition)

ZS Milling top edge—face (replaces the reference plane from the contour definition)

I Oversize in X direction (radius)

K Oversize in Z direction

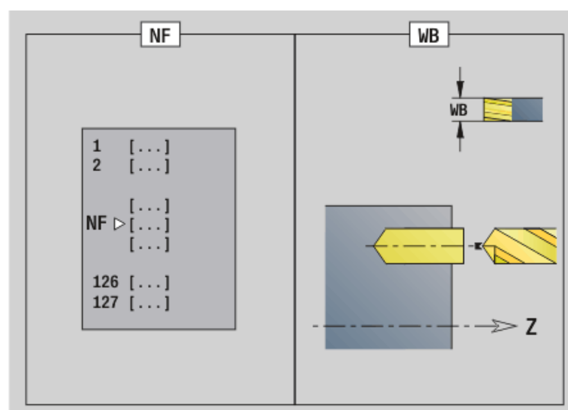
Q Machining direction (default: 0)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

A Sequence for "Calculate hole positions": A=1

NF Position mark—reference at which the cycle stores the hole positions [1 to 127].

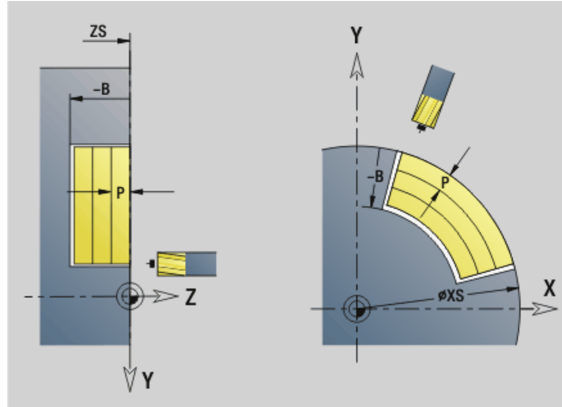
WB (Plunging length) Diameter of the milling cutter



- G845 overwrites any hole positions that may still be stored at the reference "NF."
- The parameter "WB" is used both for calculating the hole positions and for milling. When calculating the hole positions, "WB" describes the diameter of the milling cutter.

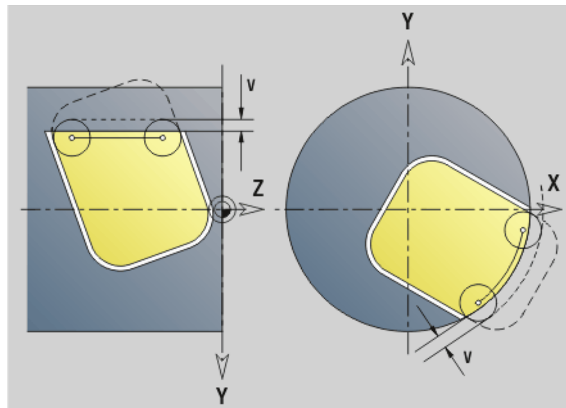
G845 (Y axis)—Milling

You can change the **cutting direction** with the "cutting direction H," the "machining direction Q" and the direction of tool rotation (see table G845 in the User's Manual). Program only the parameters given in the following table.



See also:

- G845—Fundamentals: Page 508
- G845—Calculating hole positions: Page 509



Parameters—Milling

ID Milling contour—name of the contour to be milled

NS Starting block no. of contour

- Figures: Block number of the figure
- Free closed contour: A contour element (not starting point)

B Milling depth (default: depth from the contour description)

P Maximum infeed (default: milling in one infeed)

XS Milling top edge in YZ plane (replaces the reference diameter from the contour description)

ZS Milling top edge in XY plane (replaces the reference plane from the contour description)

I Oversize in X direction (radius)

K Oversize in Z direction

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

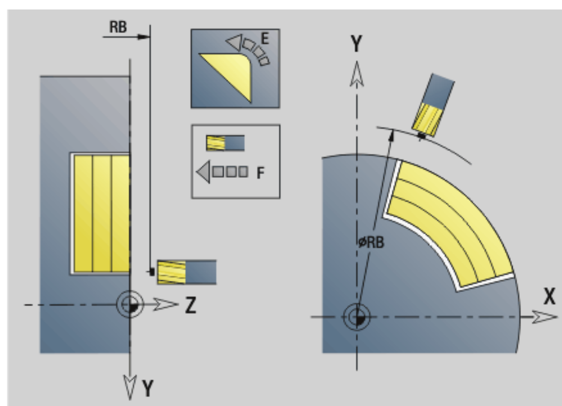
Overlap = U*milling diameter

V Overrun factor (default: 0.5). Defines the distance by which the tool should pass the outside radius of the workpiece.

- 0: The defined contour is milled completely
- 0< V <= 1: Overrun = V*milling diameter

H Cutting direction (default: 0)

- 0: Up-cut milling
- 1: Climb milling



F Feed rate for infeed (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

RB Retraction plane (default: back to starting position)

- XY plane: Retraction position in Z direction
- YZ plane: Retraction position in X direction (diameter)

Q Machining direction (default: 0)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

A Sequence for "Milling": A=0 (default=0)

NF Position mark—reference from which the cycle reads the hole positions [1 to 127].

Parameters—Milling

O Plunging behavior (default: 0)

O=0 (vertical plunge): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the pocket.

O=1 (plunge at pre-drilled position):

- If "NF" is programmed: The cycle positions the milling cutter above the first pre-drilled hole; the tool plunges and mills the first area. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next area, etc.
- If "NF" is not programmed: The tool plunges at the current position and mills the area. If applicable, position the tool to the next pre-drilled hole and mill the next area, etc.

O=2, 3 (helical plunge): The tool plunges at the angle "W" and mills full circles with the diameter "WB." As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=2—manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=3—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas.

O=4, 5 (reciprocating linear plunge): The tool plunges at the angle "W" and mills a linear path of the length "WB." You can define the orientation angle in "WE." The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=4—manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=5—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas. The plunging position is determined from the type of figure and from "Q" as follows:
  - Q0 (from the inside toward the outside):
    – Linear slot, rectangle, polygon: Reference point of the figure
    – Circle: Circle center
    – Circular slot, "free" contour: Starting point of the innermost milling path
  - Q1 (from the outside toward the inside):
    – Linear slot: Starting point of the slot
    – Circular slot, circle: Not machined
    – Rectangle, polygon: Starting point of the first linear element
    – "Free" contour: Starting point of the first linear element (at least one linear element must exist)

Parameters—Milling

O=6, 7 (reciprocating circular plunge): The tool plunges at the plunging angle "W" and mills a circular arc of 90°. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling. "WE" defines the arc center, "WB" the arc radius.

- O=6—manually: The tool position corresponds to the center of the circular arc. The tool moves to the arc starting point and plunges.
- O=7—automatically (only permitted for circular slots and circles): The cycle calculates the plunging position on the basis of "Q":
  - Q0 (from the inside toward the outside):
    – Circular slot: The circular arc lies on the curvature radius of the slot
    – Circle: Not permitted
  - Q1 (from the outside toward the inside): Circular slot, circle: The circular arc lies on the outermost milling path

W Plunging angle in infeed direction

WE Orientation angle of the milling path/circular arc. Reference axis:

- Front or rear face: Positive XK axis
- Lateral surface: Positive Z axis

Default orientation angle, depending on "O":

- O=4: WE= 0°
- O=5 and
  - Linear slot, rectangle, polygon: WE= position angle of the figure
  - Circular slot, circle: WE=0°
  - "Free" contour and Q0 (from the inside toward the outside): WE=0°
  - "Free" contour and Q1 (from the outside toward the inside): Orientation angle of the starting element

WB Plunge length/plunge diameter (default: 1.5 * milling diameter)

For the cutting direction, machining direction and direction of tool rotation, please refer to table G845 in the User's Manual.

For the machining direction Q=1 (from the outside toward the inside), please note:

- The contour must start with a linear element.
- If the starting element is < WB, WB is reduced to the length of the starting element.
- The length of the starting element must not be less than 1.5 times the diameter of the milling cutter.

Cycle run

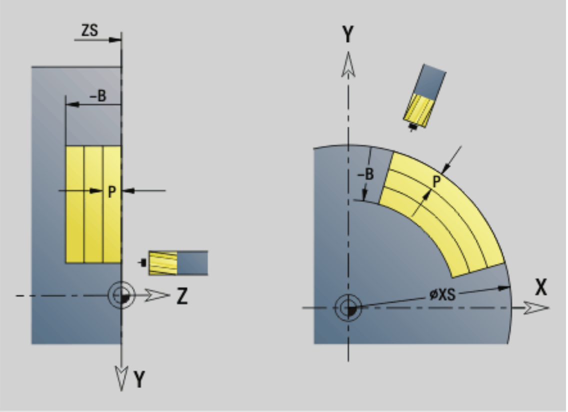
1. Starting position (X, Y, Z, C) is the position before the cycle begins.
2. Calculates the number of cuts (infeeds to the milling planes, infeeds in the milling depths) and the plunging positions and paths for reciprocating or helical plunges.
3. Approaches to safety clearance and, depending on O, feeds to the first milling depth or approaches helically or on a reciprocating path.
4. Mills a plane.
5. Retracts by the safety clearance, returns and cuts to the next milling depth.
6. Repeat steps 4 and 5 until the complete surface is milled.
7. Returns to retraction plane RB.

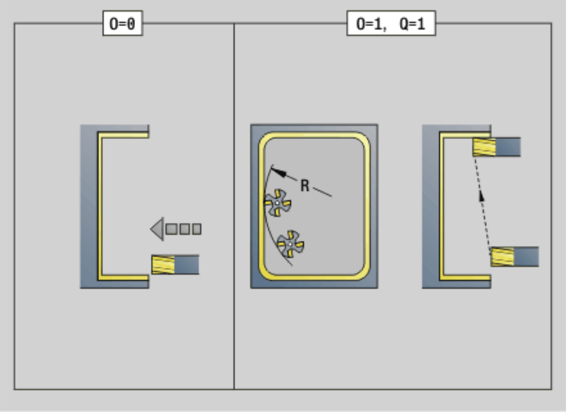
Pocket milling—finishing G846 (Y axis)

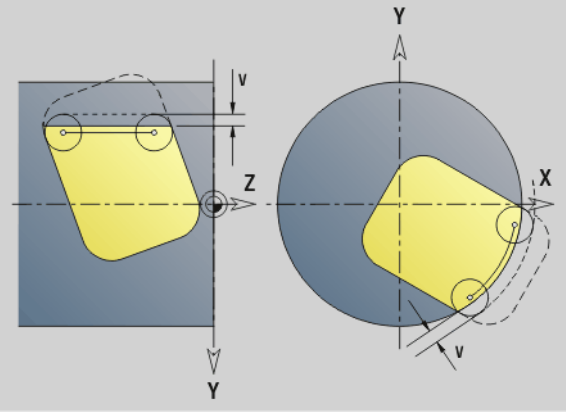
G846 finishes closed contours defined in the XY or YZ plane in the program sections:

- FACE_Y
- REAR SIDE_Y
- LATERAL_Y

You can change the **cutting direction** with the "cutting direction H," the "machining direction Q" and the direction of tool rotation.







Parameters—finishing

ID Milling contour—name of the contour to be milled

NS Starting block no. of contour

- Figures: Block number of the figure
- Free closed contour: A contour element (not starting point)

B Milling depth (default: depth from the contour description)

P Maximum infeed (default: milling in one infeed)

XS Milling top edge in YZ plane (replaces the reference diameter from the contour description)

ZS Milling top edge in XY plane (replaces the reference plane from the contour description)

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly. Feed to the starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

V Overrun factor—no effect with C-axis machining

H Cutting direction (default: 0)

- 0: Up-cut milling
- 1: Climb milling

F Feed rate for infeed (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

RB Retraction plane (default: back to starting position)

- XY plane: Retraction position in Z direction
- YZ plane: Retraction position in X direction (diameter)

Q Machining direction (default: 0)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

Parameters—finishing

O Plunging behavior (default: 0)

- O=0 (vertical plunge): The cycle moves the tool to the starting point; the tool plunges and finishes the pocket.
- Q=1 (Approaching arc with depth feed): When machining the upper milling planes, the tool advances to the milling plane and then approaches on an arc. When machining the bottom milling plane, the tool plunges to the milling depth while moving on the approaching arc (three-dimensional approaching arc). You can use this approach behavior only in conjunction with an approaching arc "R" and when machining from the outside toward the inside (Q=1).

For the cutting direction, machining direction and direction of tool rotation, please refer to table G846 in the User's Manual.

Cycle run

1. Starting position (X, Y, Z, C) is the position before the cycle begins.
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths).
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Return to "retraction plane J."

Engraving in XY plane G803

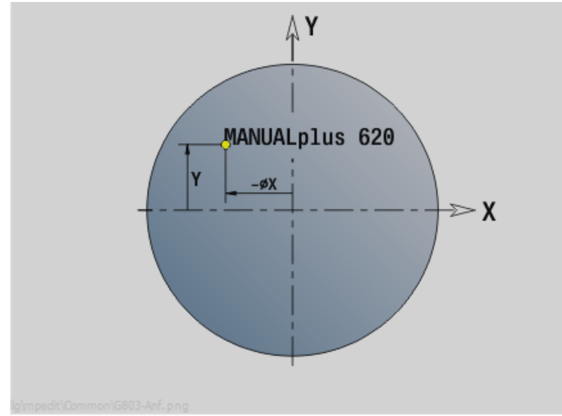
G803 engraves character strings aligned linearly in the XY plane. Character set: see page 366

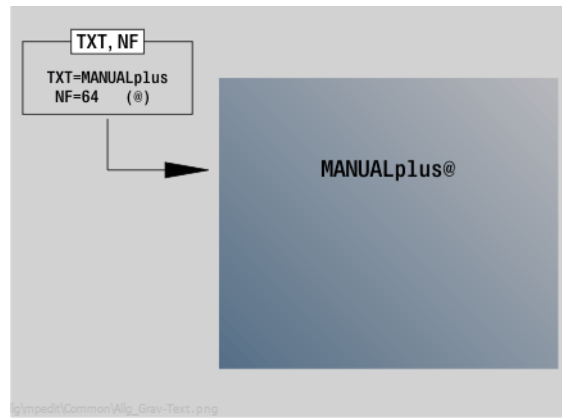
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

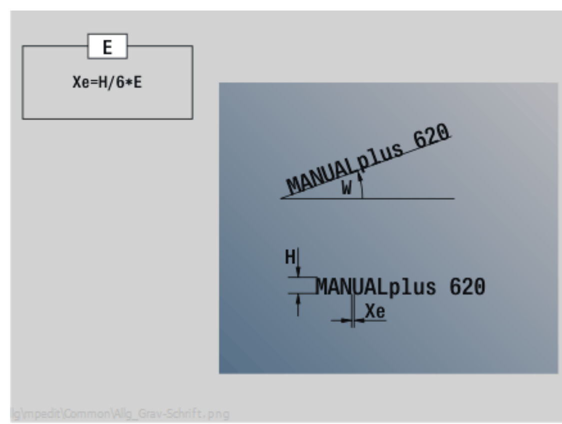
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Parameters

X, Y	Start point
Z	End point. Z position, infeed depth during milling.
RB	Retraction plane. Z position retracted to for positioning.
ID	Text to be engraved
NF	Character number (character to be engraved)
W	Orientation angle of the character string. Example: 0° = Vertical characters: the characters are aligned in sequence in positive X direction
H	Font height
E	Distance factor (for calculation see figure)
F	Plunging feed rate factor (plunging feed rate = current feed rate * F)







Engraving in the YZ plane G804

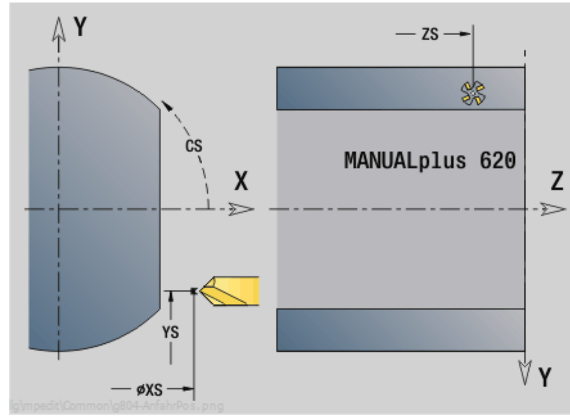
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

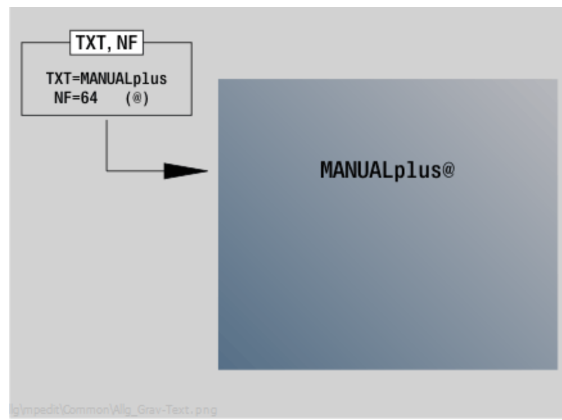
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

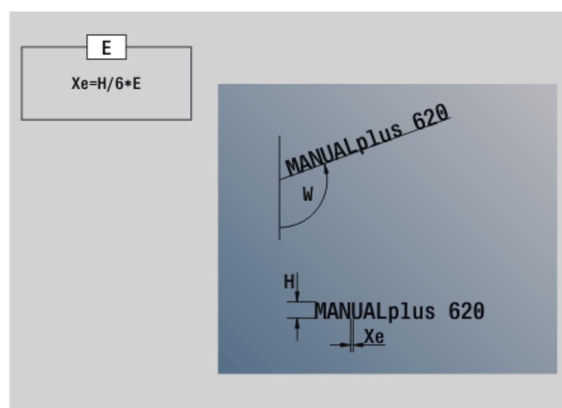
G804 engraves character strings aligned linearly in the YZ plane. Character set: see page 366

Parameters

- Y, Z Start point
- X Final point (diameter). X position, infeed depth during milling.
- RB Retraction plane. X position retracted to for positioning.
- ID Text to be engraved
- NF Character number. ASCII code of the character to be engraved
- H Font height
- E Distance factor (for calculation see figure)
- E Distance factor. The distance between the characters is calculated according to the following formula: H / 6 * E
- F Plunging feed rate factor (plunging feed rate = current feed rate * F)



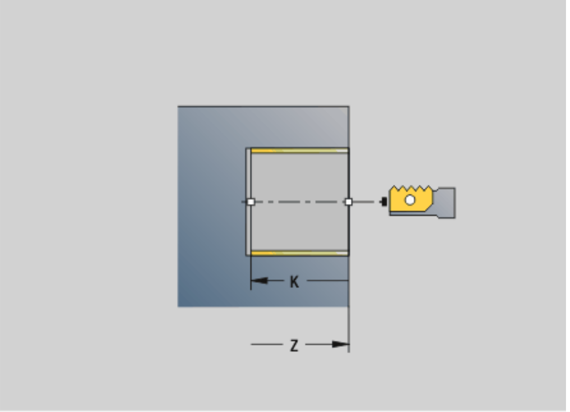


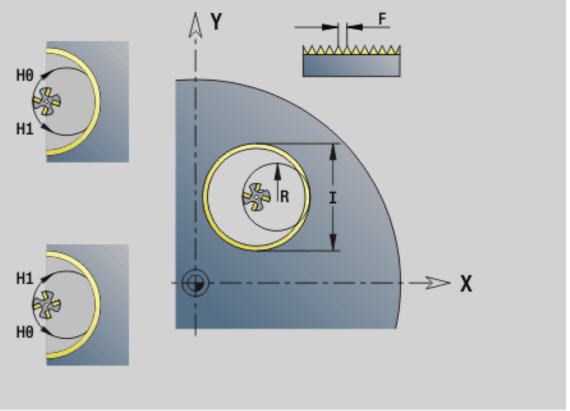


Thread milling in XY plane G800

G800 mills a thread in existing holes.

Place the tool on the center of the hole before calling G799. The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one rotation or, with single-point tools, in several rotations.





Parameters

I	Thread diameter
Z	Starting point Z
K	Thread depth
R	Approach radius
F	Thread pitch
J	Direction of thread—(default: 0) ■ 0: Right-hand thread ■ 1: Left-hand thread
H	Cutting direction (default: 0) ■ 0: Up-cut milling ■ 1: Climb milling
V	Milling method ■ 0: The thread is milled in a 360-degree helix ■ 1: The thread is milled in several helical paths (single-point tool)

Use thread-milling tools for cycle G800.

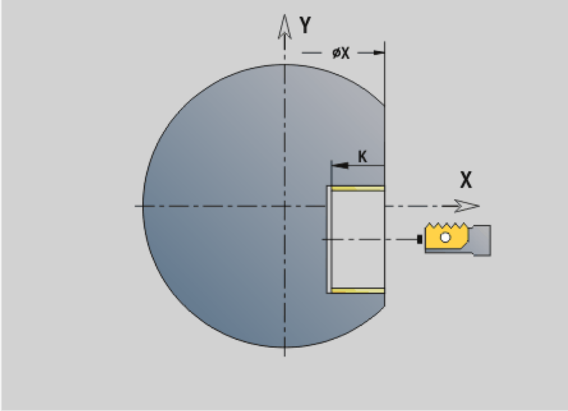
Danger of collision!

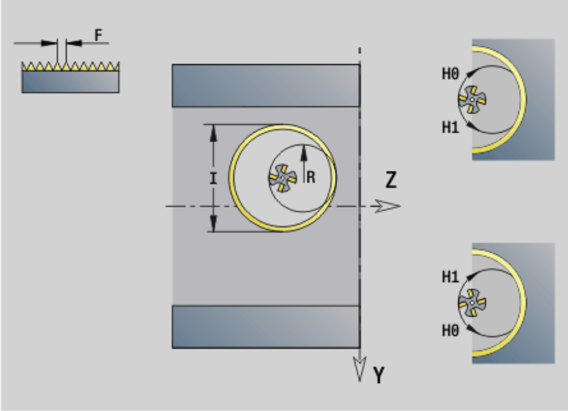
Be sure to consider the hole diameter and the diameter of the milling cutter when programming "approach radius R."

Thread milling in YZ plane G806

G806 mills a thread in existing holes.

Place the tool on the center of the hole before calling G799. The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one rotation or, with single-point tools, in several rotations.





Parameters

- I Thread diameter
- X Starting point X
- K Thread depth
- R Approach radius
- F Thread pitch
- J Direction of thread—(default: 0)
  - 0: Right-hand thread
  - 1: Left-hand thread
- H Cutting direction (default: 0)
  - 0: Up-cut milling
  - 1: Climb milling
- V Milling method
  - 0: The thread is milled in a 360-degree helix
  - 1: The thread is milled in several helical paths (single-point tool)

Use thread-milling tools for cycle G806.

Danger of collision!

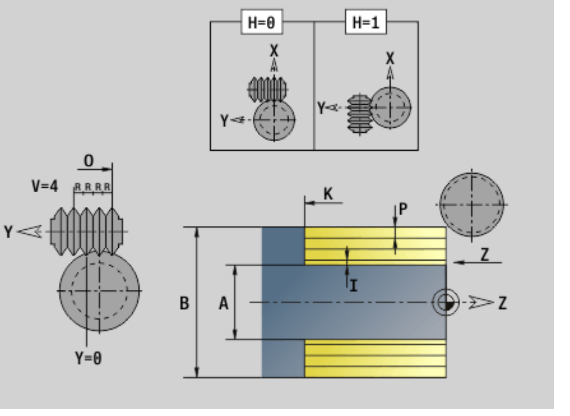
Be sure to consider the hole diameter and the diameter of the milling cutter when programming "approach radius R."

Hobbing G808

G808 mills a gear profile from the "starting point in Z" to the "end point K". In W you enter the angular position of the tool.

If an oversize has been programmed, hobbing is split up in rough-machining and subsequent finishing.

In parameters O, R and V you define the tool shift. Shifting by R ensures a uniform wear of the hob cutter.



Parameters

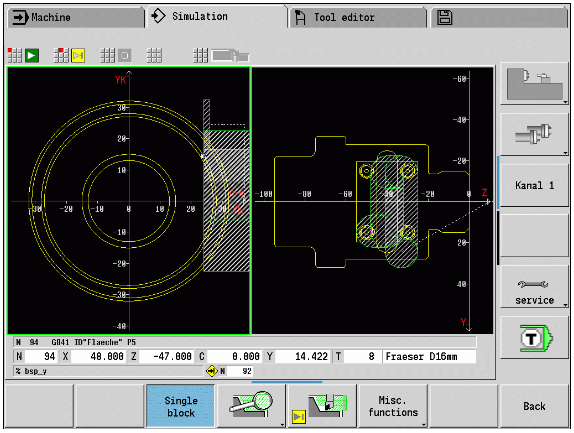
- Z Starting point
- K End point
- A Root circle diameter
- B Outside diameter
- J Number of teeth, workpiece
- W Angular position
- S Surface speed [m/min]
- I Oversize
- D Direction of rotation of the workpiece
  - 3: M3
  - 4: M4
- F Feed per revolution
- E Finishing feed rate
- P Maximum infeed
- O Shift starting position
- R Shift value
- V Number of shifts
- H Infeed axis
  - 0: Tool infeed is performed in the X axis
  - 1: Tool infeed is performed in the Y axis
- Q Workpiece spindle
  - 0: Spindle no. 0 (main spindle) holds the workpiece
  - 3: Spindle no. 3 (opposing spindle) holds the workpiece

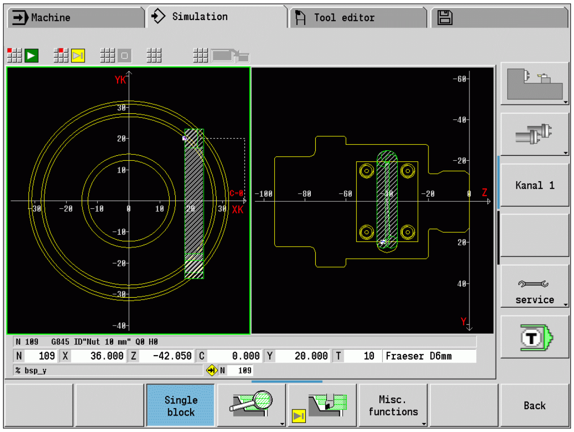
6.8 Example program

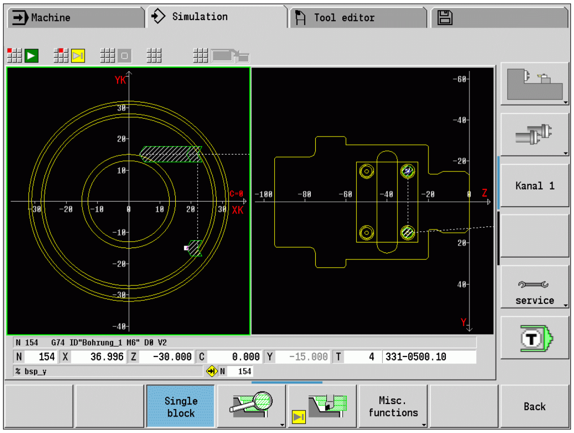
Machining with the Y axis

The milling and drilling contours are nested in the following NC program. A linear slot is machined on the single surface. On the same single surface, a hole pattern with two holes is machined both to the left and right of the slot.

At first, the turning operation is performed, and then the single surface is milled. Following that, the linear slot is machined using the "Pocket milling, lateral surface Y" unit. Then the slot is deburred. Further units are used to center the hole patterns, then drill them and finally tap the holes.







Example: "Y axis [BSP_Y.NC]"

HEADER	
#MATERIAL Aluminum	
#WORKPIECE Example Y axis	
#UNIT Metric	
TURRET 1	
T1 ID"Roughing 80 G."	
T2 ID"NC center drill"	
T3 ID"Finishing 35 G."	
T4 ID"Drill 5.2mm"	
T5 ID"Thread outside"	
T6 ID"Tapping M6".	
T8 ID"Mill D16mm"	
T10 ID"Mill D6mm"	
T12 ID"Deburring_m"	
BLANK	
N 1 G20 X70 Z97 K1	
FINISHED PART	
N 2 G0 X0 Z0	
N 3 G1 X30 BR-2	
N 4 G1 Z-20	
N 5 G25 H7 I1.5 K7 R1 W30 FP2	[Undercut DIN 76]
N 6 G1 X56 BR-1	
N 7 G1 Z-60	
N 8 G1 X64 BR-1	
N 9 G1 Z-75 BR-1	
N 10 G1 X44 BR3	
N 11 G1 Z-95 BR-1	
N 12 G1 X0	
N 13 G1 Z0	
LATERAL_Y X56 C0	[Define YZ plane]
N 14 G308 ID"Surface"	
N 15 G386 Z-55 Ki8 B30 X56 C0	[Single surface]
N 16 G308 ID"Slot 10mm" P-2	
N 17 G381 Z-40 Y0 A90 K50 B10	[Linear slot on single surface]

N 18 G309	
N 19 G308 ID"Hole_1 M6" P-15	
N 20 G481 Q2 Z-30 Y15 K-30 J-15	[Linear pattern on single surface]
N 21 G380 B5.2 P15 W118 I6 J10 F1 V0 o7	[Drilling, tapping, centering]
N 22 G309	
N 23 G308 ID"Hole_2 M6" P-15	
N 24 G481 Q2 Z-50 Y15 K-50 J-15	[Linear pattern on single surface]
N 25 G380 B5.2 P15 W118 I6 J10 F1 V0 O7	[Drilling, tapping, centering]
N 26 G309	
N 27 G309	
MACHINING	
N 28 UNIT ID"START"	[Start of program]
N 30 G26 S3500	
N 31 G126 S2000	
N 32 G59 Z256	
N 33 G140 D1 X400 Y0 Z500	
N 34 G14 Q0 D1	
N 35 END_OF_UNIT	
N 36 UNIT ID"G820_ICP"	[G820 Transverse roughing, ICP]
N 38 T1	
N 39 G96 S220 G95 F0.35 M3	
N 40 M8	
N 41 G0 X72 Z2	
N 42 G47 P2	
N 43 G820 NS3 NE3 P2 I0 K0 H0 Q0 V3 D0	
N 44 G47 M9	
N 45 END_OF_UNIT	
N 46 UNIT ID"G810_ICP"	[G810 Longitudinal roughing, ICP]
N 48 T1	
N 49 G96 S220 G95 F0.35 M3	
N 50 M8	
N 51 G0 X72 Z2	
N 52 G47 P2	
N 53 G810 NS4 NE9 P3 I0.5 K0.2 H0 Q0 V0 D0	
N 54 G14 Q0 D1	

N 55 G47 M9	
N 56 END_OF_UNIT	
N 57 UNIT ID"G890_ICP"	[G890 Contouring in ICP]
N 59 T3	
N 60 G96 S260 G95 F0.18 M4	
N 61 M8	
N 62 G0 X72 Z2	
N 63 G47 P2	
N 64 G890 NS4 NE9 V1 Q0 H3 O0 B0	
N 65 G14 Q0 D1	
N 66 G47 M9	
N 67 END_OF_UNIT	
N 68 UNIT ID"G32_MAN"	[G32 Cylindrical thread, direct]
N 70 T5	
N 71 G97 S800 M3	
N 72 M8	
N 73 G0 X30 Z5	
N 74 G47 P2	
N 75 G32 X30 Z-19 F1.5 BD0 IC8 H0 V0	
N 76 G14 Q0 D1	
N 77 G47 M9	
N 78 END_OF_UNIT	
N 79 UNIT ID"C_AXIS_ON"	[C axis on]
N 81 M14	
N 82 G110 C0	
N 83 END_OF_UNIT	
N 84 UNIT ID"G841_Y_MANT"	[Single surface in Y axis, latrl.]
N 86 T8	
N 87 G197 S1200 G195 F0.25 M104	
N 88 M8	
N 89 G19	
N 90 G110 C0	
N 91 G0 Y0	
N 92 G0 X74 Z10	

N 93 G147 K2 I2	
N 94 G841 ID"Surface" P5	[Mill a single surface]
N 95 G47 M9	
N 96 G14 Q0 D1	
N 97 G18	
N 98 END_OF_UNIT	
N 99 UNIT ID"G845_TAS_Y_MANT"	[ICP pocket mill, lateral surf. Y]
N 101 T10	
N 102 G197 S1200 G195 F0.18 M104	
N 103 G19	
N 104 M8	
N 105 G110 C0	
N 106 G0 Y0	
N 107 G0 X74 Z-40	
N 108 G147 I2 K2	
N 109 G845 ID"Slot 10 mm" Q0 H0	[Mill a slot on single surface]
N 110 G47 M9	
N 111 G14 Q0 D1	
N 112 G18	
N 113 END_OF_UNIT	
N 114 UNIT ID"G840_ENT_Y_MANT"	[ICP deburring, lateral surf. Y]
N 116 T12	
N 117 G197 S800 G195 F0.12 M104	
N 118 G19	
N 119 M8	
N 120 G110 C0	
N 121 G0 Y0	
N 122 G0 X74 Z-40	
N 123 G147 I2 K2	
N 124 G840 ID"Slot 10mm" Q1 H0 P0.8 B0.15	[Deburr slot on single surface]
N 125 G47 M9	
N 126 G14 Q0 D1	
N 127 G18	
N 128 END_OF_UNIT	
N 129 UNIT ID"G72_ICP_Y"	[ICP boring, countersinking in Y]

N 131 T2	
N 132 G197 S1000 G195 F0.22 M104	
N 133 M8	
N 134 G147 K2	
N 135 G72 ID"Hole_1 M6" D0	[Center the holes of the first pattern]
N 136 G47 M9	
N 137 END_OF_UNIT	
N 138 UNIT ID"G72_ICP_Y"	[ICP boring, countersinking in Y]
N 140 T2	
N 141 G197 S1000 G195 F0.22 M104	
N 142 M8	
N 143 G147 K2	
N 144 G72 ID"Hole_2 M6" D0	[Center the holes of the second pattern]
N 145 G47 M9	
N 146 G14 Q0 D1	
N 147 END_OF_UNIT	
N 148 UNIT ID"G74_ICP_Y"	[ICP drilling in Y axis]
N 150 T4	
N 151 G197 S1200 G195 F0.24 M103	
N 152 M8	
N 153 G147 K2	
N 154 G74 ID"Hole_1 M6" D0 V2	[Holes of the first pattern]
N 155 G47 M9	
N 156 END_OF_UNIT	
N 157 UNIT ID"G74_ICP_Y"	[ICP drilling in Y axis]
N 159 T4	
N 160 G197 S1200 G195 F0.24 M103	
N 161 M8	
N 162 G147 K2	
N 163 G74 ID"Hole_2 M6" D0 V2	[Holes of the second pattern]
N 164 G47 M9	
N 165 G14 Q0 D1	
N 166 END_OF_UNIT	
N 167 UNIT ID"G73_ICP_Y"	[ICP tapping in Y axis]

N 169 T6	
N 170 G197 S800 M103	
N 171 M8	
N 172 G147 K2	
N 173 G73 ID"Hole_1 M6" F1	[Tapping, first pattern]
N 174 G47 M9	
N 175 END_OF_UNIT	
N 176 UNIT ID"G73_ICP_Y"	[ICP tapping in Y axis]
N 178 T6	
N 179 G197 S800 M103	
N 180 M8	
N 181 G147 K2	
N 182 G73 ID"Hole_2 M6" F1	[Tapping, second pattern]
N 183 G47 M9	
N 184 G14 Q0 D1	
N 185 END_OF_UNIT	
N 186 UNIT ID"C_AXIS_OFF"	[C axis off]
N 188 M15	
N 189 END_OF_UNIT	
N 190 UNIT ID"END"	[Program end]
N 192 M30	
N 193 END_OF_UNIT	
END	

7

TURN PLUS

7.1 TURN PLUS mode of operation

To create programs with TURN PLUS, you program the blank and finished part with the aid of interactive graphics. The working plan is then generated automatically. As a result you get a commented and structured NC program.

With TURN PLUS you can create NC programs for the following applications:

- Turning operations
- Drilling and milling with the C axis
- Drilling and milling with the Y axis

TURN PLUS concept

The workpiece description is the basis for working plan generation. The strategy for generating the working plan is specified in the **machining sequence.** The **machining parameters** define details of machining. This allows you to adapt TURN PLUS to your individual needs.

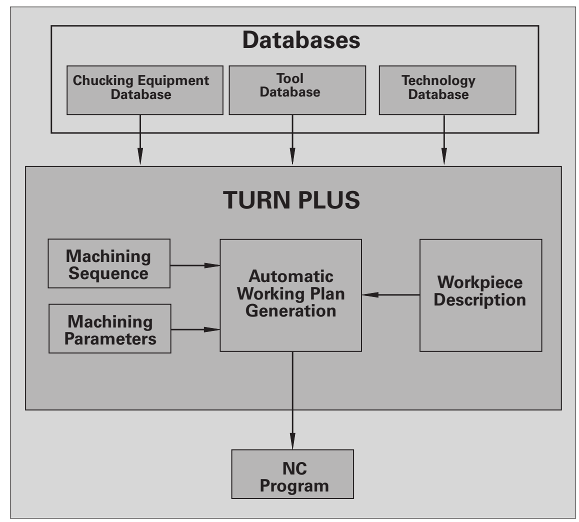
TURN PLUS generates the working plan, which takes technology attributes such as oversizes, tolerances, etc. into account.

On the basis of the **blank form update**, TURN PLUS optimizes the paths for approach and avoids air cuts or collisions between workpiece and cutting edge.

TURN PLUS uses the current turret assignment for the tool selection. If there is no suitable tool in the turret assignment, TURN PLUS looks for a suitable tool in the tool database.

When clamping the workpiece, TURN PLUS determines the cutting limitations and the zero point shift for the NC program.

The technology database provides the cutting data to TURN PLUS.



7.2 Automatic working plan generation (AWG)

The **AWG** generates the work blocks of the working plan in the sequence defined in "Machining sequence." You define the machining details in the **Machining Parameters** input form. TURN PLUS automatically finds all the elements of a work block. Use the **machining sequence editor** to specify the machining sequence.

A work block has the following content:

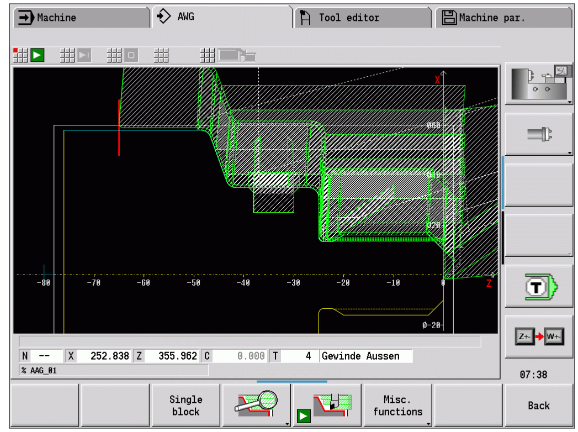
- Tool call
- Cutting values (technology data)
- Approach (may be omitted)
- Machining cycle
- Tool retraction (may be omitted)
- Moving to tool change point (may be omitted)

You can change or supplement the generated work blocks subsequently.

TURN PLUS simulates the machining in the AWG control graphic. You can set the sequence and representation of the control graphic via soft key (see "Graphic simulation" in the User's Manual).

> TURN PLUS outputs warnings during the contour analysis if certain areas cannot be machined at all or not completely. Check the respective sections after program creation and adapt them to your needs.

Generating a working plan



After the working plan generation, consider the following: If no chucking equipment has been defined in the program as yet, TURN PLUS defines the chucking equipment for a specific type of clamping/clamping length and adjusts the cutting limitation accordingly. Adapt the values in the finished NC program.

Generating a working plan with TURN PLUS

Select "TURN PLUS." TURN PLUS opens the most recently selected machining sequence.

AWG — Select "AWG." TURN PLUS shows the contours of the blank and the finished part in the graphics window.

Change window — Press the "AWG control graphic" soft key: The AWG control graphic and program generation are started.

Back — Press the "Back" soft key to return to the TURN PLUS menu.

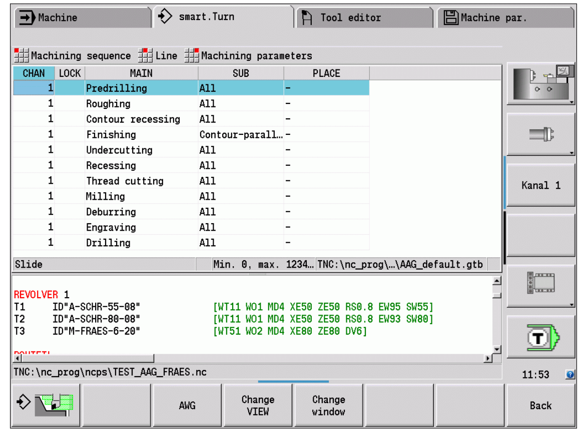
Back — Press the "Back" soft key to switch to smart.Turn.

Save — Use the name of the current program without any changes and press the "Save" soft key to overwrite the current program.

Save — Enter a name for the program and confirm with the the "Save" soft key.

Machining sequence – Fundamentals

TURN PLUS analyzes the contour in the sequence defined in "Machining sequence." In this process the areas to be machined and the tool parameters are ascertained. The AWG analyzes the contour using the machining parameters.



TURN PLUS distinguishes between:

- Main machining operation (e.g. Undercutting)
- Submachining operation (e.g. type H, K, or U)
- Machining location (e.g. outside or inside)

"Submachining" and "machining location" refine the machining specification. If you do not define the submachining operation or machining location, the AWG generates machining blocks for **all** submachining operations/machining locations.

The following factors additionally influence the working plan generation:

- Geometry of the contour
- Attributes of the contour
- Tool availability
- Machining parameters

In the machining sequence you define the sequence in which the machining steps are carried out. If you only define the main machining operation in the sequence for a machining operation, all of the submachining operations comprised by it are executed in a specific sequence. However, you can also program the submachining operations and machining locations individually in any sequence. In this case you should define the associated main machining operation again after defining the submachining operations. This way you can ensure that all submachining operations and locations are taken into account.

The machining sequence and the program can be displayed in a horizontal or vertical window layout. Press the "Change VIEW" soft key to switch between the two views.

Press the "Change window" soft key to move the cursor between the Program and the Machining Sequence window.

The AWG does **not** generate the work blocks if any required preparatory step is missing, or if the appropriate tool is not available, etc. TURN PLUS skips machining operations/machining sequences that do not make sense in the machining process.

Organizing machining sequences:

- TURN PLUS always uses the **current machining sequence**. The current machining sequence can be edited or overwritten by loading another machining sequence.
- When you open TURN PLUS, the most recently used machining sequence is automatically displayed.

Danger of collision!

When executing drilling or milling operations, TURN PLUS does not check whether the turning operation has already been completed. Ensure that turning operations precede drilling or milling operations in the machining sequence.

Editing and managing machining sequences

TURN PLUS uses the currently active machining sequence. You can change the machining sequences and adapt them to your range of parts.

Managing the machining sequence files:

Open machining sequence:

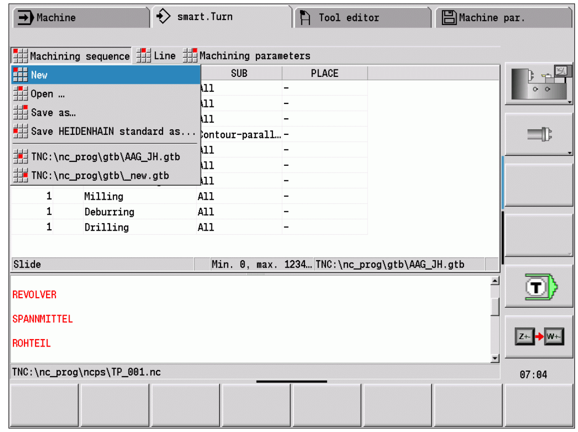
- Select "TURN PLUS > Machining sequence > Open." TURN PLUS opens the selection list with the machining sequence files.
- Select the desired file.

To save the machining sequence:

- Select "TURN PLUS > Machining sequence > Save as." TURN PLUS opens the selection list with the machining sequence files.
- Enter a new file name or overwrite an existing file.

To create a default machining sequence:

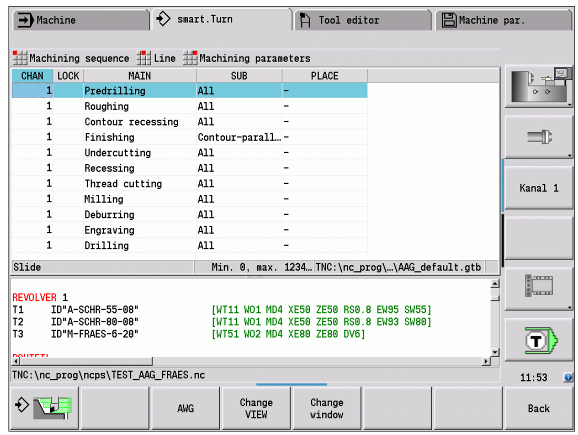
- Select "TURN PLUS > Maching sequence >Save HEIDENHAIN standard as..." TURN PLUS opens the selection list with the machining sequence files.
- Enter a file name under which you wish to store the HEIDENHAIN default machining sequence.



Editing a machining sequence

Position the cursor

Select "TURN PLUS > Machining sequence > Line." Select the function.



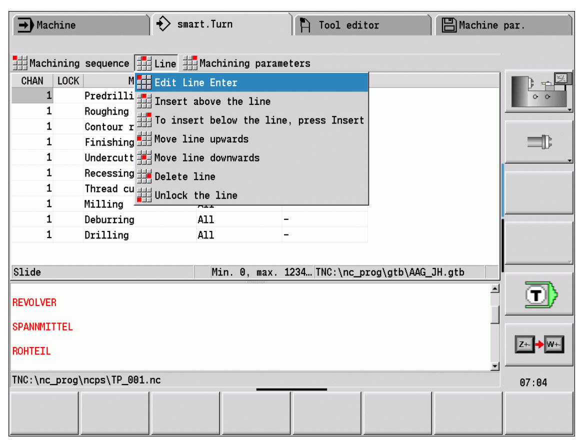
Insert new machining operation.

To insert a new machining operation before the cursor position, select "Insert above the line."

To insert a new machining operation after the cursor position, select "To insert below the line, press Insert."

Moving a machining operation

Select "Move line upwards" or "Move line downwards."



Editing the machining sequence

Select "Edit line."

"OK" confirms the new machining sequence.

Deleting a machining operation

"Delete line" deletes the selected machining sequence.

Overview of machining sequences

The following table lists the possible combinations of main machining operations with submachining operations and machining locations and explains the working method of the AWG.

Machining sequence for predrilling

Main machining	Submachining	Location	Execution
Predrilling			**Contour analysis:** Determining the drilling steps **Machining parameter:** 3 – Centric predrilling
	All	–	Predrilling

Machining sequence "Roughing"

Main machining	Submachining	Location	Description
Roughing			**Contour analysis**: Dividing the contour into areas for longitudinal/ transverse outside machining and longitudinal/transverse inside machining according to the transverse/longitudinal ratio. **Sequence**: First outside, then inside machining **Machining parameter:** 4 – Roughing
	All	–	Transverse machining, longitudinal machining – outside and inside
	Longitudinal machining	–	Longitudinal machining – outside and inside
	Longitudinal machining	Outside	Longitudinal machining – outside
	Longitudinal machining	Inside	Longitudinal machining – inside
	Transverse machining	–	Transverse machining – outside and inside

Main machining	Submachining	Location	Description
	Transverse machining	Outside	Transverse machining – outside
	Transverse machining	Inside	Transverse machining – inside
	Contour-parallel	–	Contour-parallel machining – outside and inside
	Contour-parallel	Outside	Contour-parallel machining – outside
	Contour-parallel	Inside	Contour-parallel machining – inside

Machining sequence "Finishing"

Main machining	Submachining	Location	Execution
Finishing			**Contour analysis**: Dividing the contour into areas for outside and inside machining. **Sequence**: First outside, then inside machining **Machining parameter:** 5 – Finishing
	Contour-parallel	–	Outside/inside machining
	Contour-parallel	Outside	Outside machining
	Contour-parallel	Inside	Inside machining

Machining sequence for recess turning

Main machining	Submachining	Location	Execution
Recess turning			**Contour analysis**: ■ Without previous **roughing operation**: The complete contour, including recess areas (undefined recesses), is machined. ■ With previous **roughing**: Recess areas (undefined recesses) are determined and machined according to the "inward copying angle (EKW)." **Sequence**: First outside, then inside machining **Machining parameter:** 1 Global parameters for finished parts
	All	–	Radial/axial machining – outside and inside
	Longitudinal machining	Outside	Radial machining – outside
	Longitudinal machining	Inside	Radial machining – inside
	Transverse machining	Outside/ front	Axial machining – outside
	Transverse machining	Inside/front	Axial machining – inside

Recess turning and contour turning are used alternatively.

Machining sequence for contour recessing

Main machining	Submachining	Location	Execution
Contour recessing			**Contour analysis:** Recess areas (recesses) are determined and machined according to the "inward copying angle (EKW)." **Sequence**: First outside, then inside machining **Machining parameter:** 1 Global parameters for finished parts
	All	–	Radial/axial machining – outside and inside Shaft machining: Axial machining on the outside is performed on front and back
	Longitudinal machining	Outside	Radial machining – outside
	Longitudinal machining	Inside	Radial machining – inside
	Transverse machining	Outside/ front	Axial machining – outside
	Transverse machining	Inside/front	Axial machining – inside

Recess turning and contour turning are used alternatively.

Machining sequence for recessing

Main machining	Submachining	Location	Execution
Recessing			**Contour analysis:** Determining the "Recess" form elements: ■ Type S (guarding ring – recess type S) ■ Type D (sealing ring – recess type D) ■ Type A (recess general) ■ Type FK (relief turn F) – FK is only machined using "Recessing" if the "inward copying angle (EKW) <= mtw." **Sequence**: First outside, then inside machining **Machining parameter** (with type FK): 1 Global parameters for finished parts
	All	–	All recess types; radial/axial machining; outside and inside
	Type S, D, A, FK	–	Radial/axial machining – outside and inside
	Type S, D, A, FK	Outside	Radial machining – outside
	Type S, D, A, FK	Inside	Radial machining – inside
	Type S, D, A, FK	Outside/ front	Axial machining – outside
	Type S, D, A, FK	Inside/front	Axial machining – inside

Machining sequence for undercuts

Main machining	Submachining	Location	Execution
Undercutting			**Contour analysis/machining:** Determining the "Undercuts" form elements: ■ Type H – Machining using single paths of traverse; copying tool (type 22x) ■ Type K – Machining using single paths of traverse; copying tool (type 22x) ■ Type U – Machining using single paths of traverse; recessing tool (type 15x) **Sequence**: First outside, then inside machining; first radial, then axial machining
	All	–	All recess types; outside and inside
	Type H, K, U	–	Radial/axial machining – outside and inside
	Type H, K, U	Outside	Machining – outside
	Type H, K, U	Inside	Machining – inside

Machining sequence for thread cutting

Main machining	Submachining	Location	Execution
Thread cutting			**Contour analysis:** Determining the "Thread" form elements.
			Sequence: First outside, then inside machining; the elements are then machined according to the sequence of geometrical definition.
	All	–	Machining cylindric (longitudinal), tapered and transverse threads on the outside and inside of a contour.
	All	Outside	Machining cylindric (longitudinal), tapered and transverse threads on the outside of a contour.
	All	Inside	Machining cylindric (longitudinal), tapered and transverse threads on the inside of a contour.
	Cylinder	–	Machining outside and inside threads
	Cylinder	Outside	Machining cylindric outside threads.
	Cylinder	Inside	Machining cylindric inside thread
	Transverse	–	Machining transverse threads on the outside and inside of the contour
	Transverse	Outside	Machining transverse threads on the outside.
	Transverse	Inside	Machining transverse threads on the inside.
	Taper	–	Machining taper threads on the outside and inside of the contour
	Taper	Outside	Machining taper threads on the outside.
	Taper	Inside	Machining taper threads on the inside.

Machining sequence for drilling

Main machining	Submachining	Location	Execution
Drilling			**Contour analysis:** Determining the "Hole" form elements. **Sequence – Drilling operations/drilling combinations:** ■ Centering / Centering and countersinking ■ Drilling ■ Countersinking / Drilling and countersinking ■ Reaming / Drilling with reaming ■ Tapping / Drilling with thread **Sequence – Location:** ■ Centric ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	All	–	All machining operations at all machining locations
	Centering, drilling, countersinking, reaming, tapping	–	Machining at all machining locations
	Centering, drilling, countersinking, reaming, tapping	Centric	Centric machining on the face
	Centering, drilling, countersinking, reaming, tapping	Face	Machining on the face
	Centering, drilling, countersinking, reaming, tapping	Lateral	Machining on the lateral surface

Machining sequence for milling

Main machining	Submachining	Location	Execution
Milling			**Contour analysis:** Determining the milling contours. **Sequence – Milling operation:** ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and centric polygons **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	All	–	All milling operations at all machining locations
	Surface, contour, slot, pocket	–	Milling at all machining locations
	Surface, contour, slot, pocket	Face	Milling the end face
	Surface, contour, slot, pocket	Lateral	Milling on the lateral surface

Machining sequence for deburring

Main machining	Submachining	Location	Execution
Deburring			**Contour analysis:** Determining milling contours with "Deburring" attribute. **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	All	–	All milling operations at all machining locations
	Contour, slot, pocket (*)	–	Deburr selected element at all machining locations
	Contour, slot, pocket (*)	Face	Deburr selected element on the face
	Contour, slot, pocket (*)	Lateral	Deburr selected element on the lateral face

*: Define the type of contour

Machining sequence for milling and finishing

Main machining	Submachining	Location	Execution
Finish-milling			**Contour analysis:** Determining the milling contours. **Sequence – Milling operation:** ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and centric polygons **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	Contour, slot, pocket (*)	–	Finish selected element at all machining locations
	Contour, slot, pocket (*)	Face	Finish selected element on the face
	Contour, slot, pocket (*)	Lateral	Finish selected element on the lateral face
	*: Define the milling operation		

Machining sequence for parting

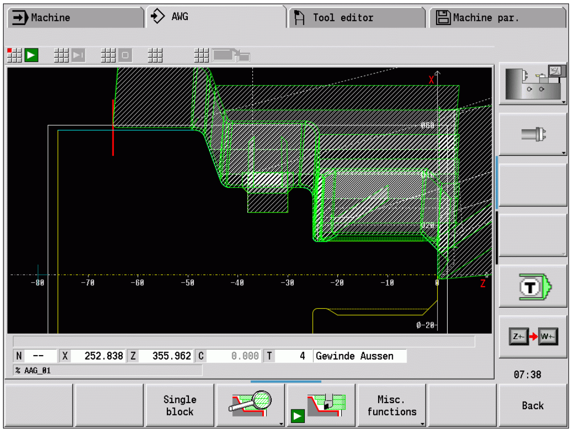
Main machining	Submachining	Location	Execution
Parting	All	–	The workpiece is cut off.

7.3 AWG control graphic

When you create a program with the **AWG**, the programmed blank and finished part are displayed in the simulation window and in addition, all machining steps are simulated successively. The workpiece blank **takes on a contour** during machining.

Setting the AWG control graphic

When you start the automatic program creation with the AWG soft key, the control automatically opens the AWG control graphic. The simulation starts dialogs in which you get machining and tool information. After you have simulated the machining process, you can close the graphics window with the "Back" soft key. The "Save as" dialog box opens once you exit the TURN PLUS menu with the "Back" soft key. The name of the opened program is displayed in the "File name" dialog field. If you do not enter another file name, the opened program will be overwritten. Alternatively, you can save the machining operation in another program.



The AWG control graphic is indicated in the soft-key symbol by a contour outlined in red.

You can set the display of the **tool paths** and the **simulation mode** as usual (see "Graphic simulation" in the User's Manual).

7.4 Machining information

Tool selection, turret assignment

The **tool selection** is determined by:

- Machining direction
- Contour to be machined
- Machining sequence

If the ideal tool is not available, TURN PLUS

- First looks for a replacement tool,
- Then for an emergency tool.

If necessary, TURN PLUS adapts the machining cycle to the requirements of the replacement or emergency tool. If more than one tool is suitable for a machining operation, TURN PLUS uses the optimal tool.

The **Mount type** distinguishes between different tool holders (see "Tool data" in the User's Manual). TURN PLUS checks whether the mount type in the tool holder description and the mount type in the turret pocket description are the same.

TURN PLUS automatically calculates the required zero point shift for the workpiece and activates it with G59. To calculate the zero point shift, TURN PLUS takes the following values into account:

- Workpiece length **Z** (description of workpiece blank)
- Oversize **K** (Description of workpiece blank)
- Edge of chuck **Z** (description of chucking equipment and machining parameters)
- Edge of chuck **B** (description of chucking equipment and machining parameters)

Contour recessing, recess turning

The **cutting radius** must be smaller than the smallest inside radius of the recess contour, but >= 0.2 mm. TURN PLUS determines the **width of the recessing tool** from the recess contour:

- Recess contour includes paraxial base elements with radii on both sides: SB <= b + 2*r (if radii differ: smallest radius).
- Recess contour includes paraxial base elements without radii or with a radius on one side: SB <= b
- Recess contour does not include paraxial base elements: The width of the recessing tool is determined from the recessing width divisor (machining parameter 6 – SBD).

Abbreviations:

- SB: Width of recessing tool
- b: Width of base element
- r: Radius

Drilling

Depending on the geometry of the bore hole, the AWG determines the appropriate tool. For centric bore holes, TURN PLUS uses stationary tools.

Cutting data, coolant

To determine the **cutting parameters**, TURN PLUS uses the

- Workpiece material (program head)
- Cutting material (tool parameters)
- The machining operation (main operation in the machining sequence).

The values determined are multiplied by the tool-dependent compensation factors (see "Tool data" in the User's Manual).

Note for roughing and finishing operations:

- Main feed rate for use of the primary cutting edge
- Auxiliary feed rate for use of the secondary cutting edge

Note for milling operations:

- Main feed rate for machining in the milling plane
- Auxiliary feed rate for infeed movements

For threading, drilling and milling operations, the cutting speed is converted into rotational speed.

Coolant: Depending on the workpiece material, cutting material and machining operation, define in the technology database whether coolant is used. The AWG activates the appropriate coolant circuits for the respective tool.

If you have specified that coolant is to be used, the AWG activates the coolant circulation for the respective machining block.

Inside contours

TURN PLUS machines continuous inside contours up to the transition from the "deepest point" to a greater diameter. The end position for drilling, roughing and finishing operations depends on:

- Cutting limit, inside
- Overhang length, inside **ULI** (Processing machining parameter)

Prerequisite: The usable tool length must be sufficient for the machining operation. If it is not, then this parameter defines the inside machining operation. The following examples explain the principle.

Limits for internal machining operations

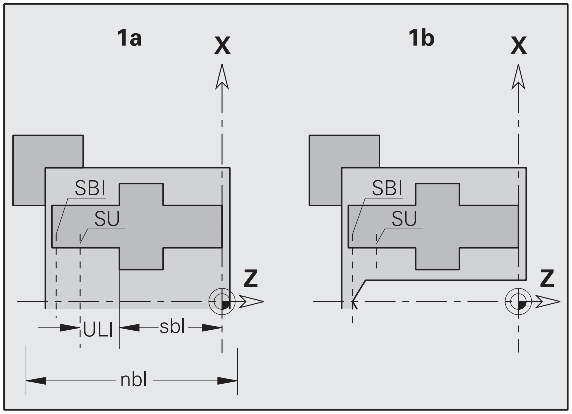
- **Predrilling: SBI** limits the drilling operation.
- **Roughing: SBI** or **SU** limit the roughing operation.
  - SU = basic length of roughing cut (sbl) + overhang length, inside (ULI)
  - To avoid residual rings during the machining process, TURN PLUS leaves an area of 5° in front of the roughing limit.
- **Finishing: sbl** limits the finishing operation.

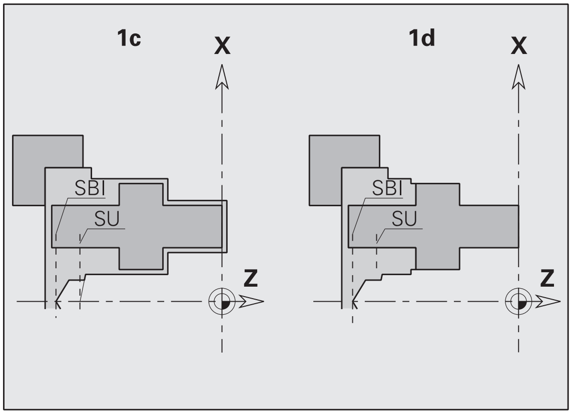
Roughing limit in front of cutting limit

Example 1: The roughing limit (SU) is located **in front of** the cutting limit, inside (SBI).

Abbreviations

- SBI: Cutting limit, inside
- SU: Roughing limitation (SU = sbl + ULI)
- sbl: Basic length of roughing cut ("deepest" point of inside contour)
- ULI: Overhang length, inside (machining parameter 4)
- nbl: Usable tool length (tool parameter)



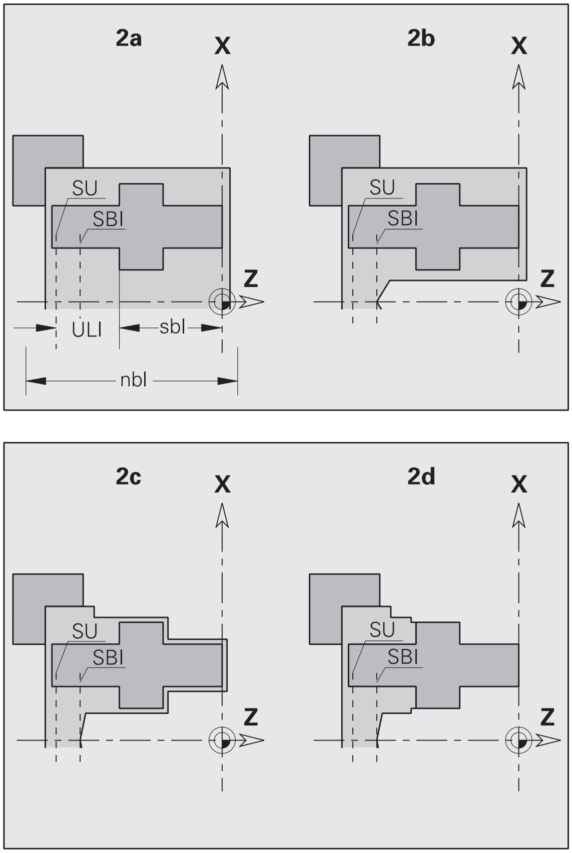


Roughing limit behind cutting limit

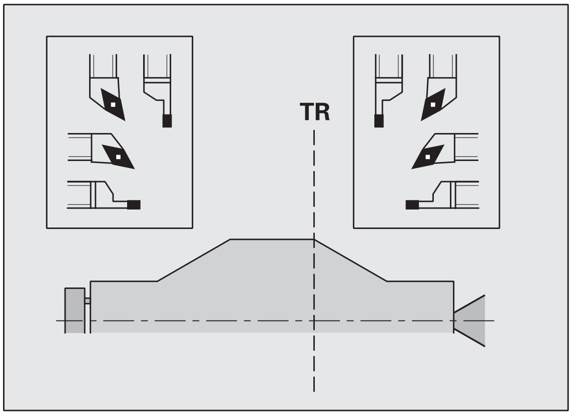
Example 2: The roughing limit (SU) is located **behind** the cutting limit, inside (SBI).

Abbreviations

- SBI: Cutting limit, inside
- SU: Roughing limitation (SU = sbl + ULI)
- sbl: Basic length of roughing cut ("deepest" point of inside contour)
- ULI: Overhang length, inside (machining parameter 4)
- nbl: Usable tool length (tool parameter)



Shaft machining



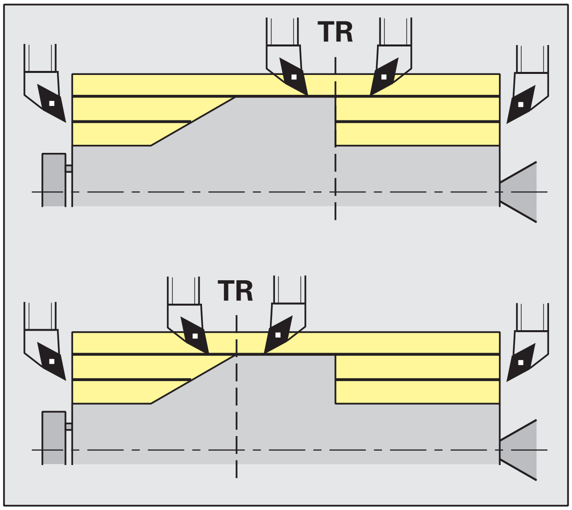
For shafts, TURN PLUS supports rear-face machining of outside contours in addition to standard machining processes. This enables shafts to be completely machined using one setup. You can select the clamping type for the shaft machining (**`Shaft/chuck`** or **`Shaft/face driver`**) in the **`V`** input parameter in the chucking equipment dialog.

TURN PLUS does **not** support retracting the tailstock and does not check the setup used.

Precondition for shaft machining: The workpiece is clamped at spindle and tailstock.

> **Danger of collision!**
>
> TURN PLUS does not monitor for collisions during transverse machining or machining operations on the end face.

Separation point (TR)



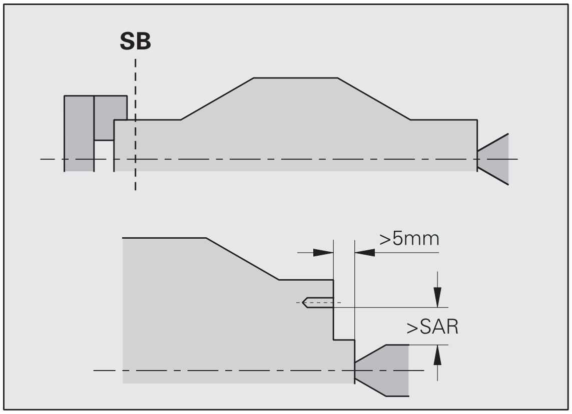
The separation point (TR) divides the workpiece into front and rear area. If no separation point has been specified, TURN PLUS sets a separation point at the transition from the largest to a smaller diameter. Position the separation points on outside corners.

Tools for machining the

- Area on front side: Main machining direction –Z; or primarily "left" recessing or tapping tools, etc.
- Area on rear side: Main machining direction + Z; or primarily "right" recessing or tapping tools, etc.

Setting/changing the separation point: See "Separation point G44" on page 215.

Protective zones for drilling and milling operations



TURN PLUS machines drilling and milling contours on transverse surfaces (front/rear face) if:

- (Horizontal) distance to transverse surface > 5 mm, or
- Distance between chucking equipment and drilling/milling contour is > SAR
  (SAR: See user parameter).

If jaws are used for clamping the shaft at the spindle, TURN PLUS accounts for the cutting limitation (O).

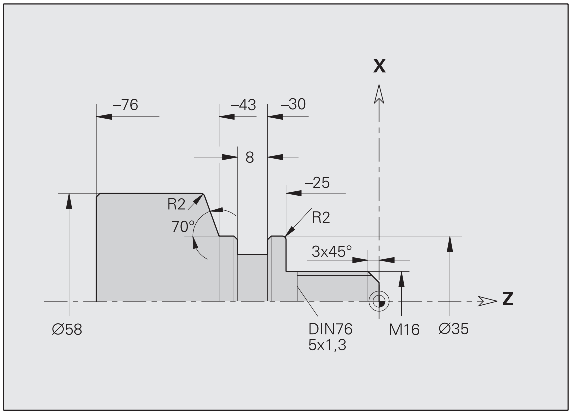
Machining information

- **Chucking the workpiece at the spindle:** Ensure that the area, where the blank part is chucked, is premachined. Otherwise, the cutting limitation might adversely affect the machining strategies.
- **Machining of bars:** TURN PLUS **does not control** the bar loader and does not move the tailstock and steady rest components. TURN PLUS does not support workpiece adjustment between collet and dead center during machining operations.
- **Transverse machining**
  - Please note that the entries made in the machining sequence apply to the complete workpiece and thus also to the transverse machining of shaft ends.
  - The AWG does not machine inside areas on the rear face. If jaws are used for clamping the shaft at the spindle, the rear face is not machined.
- **Longitudinal machining:** First the front area is machined, then the rear area.
- **Collision prevention:** If machining operations are not performed **without collisions**, you can do the following:
  - Add a retraction of the tailstock, a positioning of the steady rest, etc. to the program.
  - Add cutting limits to the program to avoid collisions.
  - Disable automatic machining in the AWG by assigning the "Exclusion from machining" attribute or by defining a specific machining location in the machining sequence.
  - Define an oversize=0 for the workpiece blank. As a consequence, the front area is not machined (e.g. shafts cut to length and centered shafts).

7.5 Example

On the basis of the production drawing, the working steps for defining the contour of the blank and finished part, the setup procedures and automatic working plan generation are explained.

Workpiece blank: Ø60 X 80; Material: Ck 45



- Undefined chamfers: 1x45°
- Undefined radii: 1 mm

Creating a program

- Select "Program > New > New DINplus Program." The control opens the "Save as" dialog box.
- Enter a program name and press the "Save" soft key. The control opens the "Program head (short)" dialog box.
- Select a material from the fixed-word list and press the "OK" soft key.

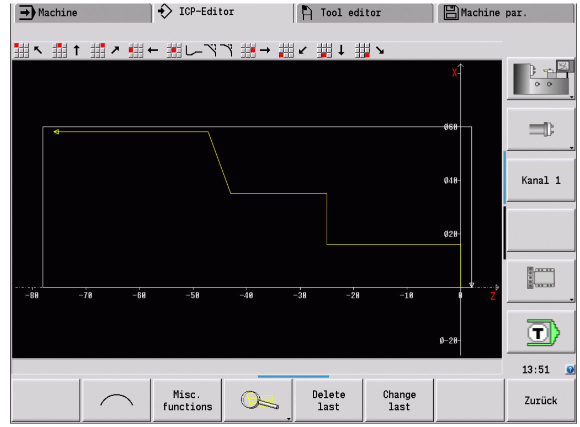
Defining the workpiece blank

- Select "ICP > Blank > Bar." TURN PLUS opens the "Bar" dialog box.
- Inputs:
  - Diameter X = 60 mm
  - Length Z = 80 mm
  - Oversize K = 2 mm
- TURN PLUS displays the workpiece blank.

- Press the "Back" soft key to return to the main menu.

Defining the basic contour

- Select "ICP > Finished part (> Contour)."
- Enter start point of the contour X = 0; Z = 0 and end point of the element X = 16
- Enter Z = –25
- Enter X = 35
- Enter Z = –43
- Enter X = 58; W = 70
- Enter Z = –76
- Press the "Back" soft key to go back one menu level.



Defining form elements

Chamfer at corner of threaded shank:

- Select the form elements.
- Select "Form > Chamfer."
- Select the corner of the threaded shank.
- "Chamfer" dialog box: Chamfer width = 3 mm

Rounding arcs:

- Select "Form > Rounding."
- Select the corners for the rounding arcs.
- "Rounding" dialog box: Rounding radius = 2 mm

Undercut:

- Select "Form > Undercut > Undercut type G."
- Select the corner for the undercut.
- "Undercut type DIN 76" dialog box

Recess:

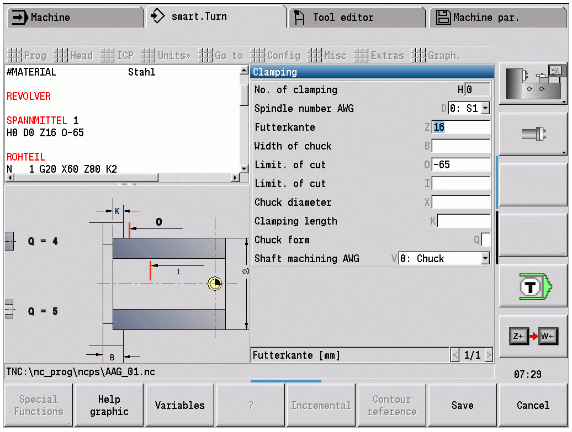
- Select "Form > Recess > Recess standard / G22."
- Select the basic element for the recess.
- "Recess standard / G22" dialog box:
  - Inside corner (Z) = 25 mm
  - Inside corner (Ki) = –8 mm
  - Recess diameter = 25 mm
  - Outside radius/chamfer (B) = –1 mm

Thread:

- Select "Form > Thread."
- Select the basic element for the thread.
- "Thread" dialog box: Select "ISO DIN 13"

Back
- Press the "Back" soft key to return to the main menu.

Preparing the machining process, chucking



TURN PLUS automatically calculates the required zero point shift for the workpiece and activates it with G59. To calculate the zero point shift, TURN PLUS takes the following values into account:

- Workpiece length **Z** (description of workpiece blank)
- Oversize **K** (Description of workpiece blank)
- Edge of chuck **Z** (description of chucking equipment and machining parameters)
- Edge of chuck **B** (description of chucking equipment and machining parameters)

- Select "Head > Chucking equipment"
- Describe the chucking equipment:
  - Select "AWG spindle number"
  - Enter the edge of chuck
  - Enter the chuck width
  - Enter the cutting limitation (outside and inside)
  - Enter the clamping diameter
  - Enter the clamping length
  - Define the clamping form
  - Select "Shaft machining AWG"
- TURN PLUS takes the chucking equipment and cutting limitation into account for the program creation.

Back
- Press the "Back" soft key to return to the main menu.

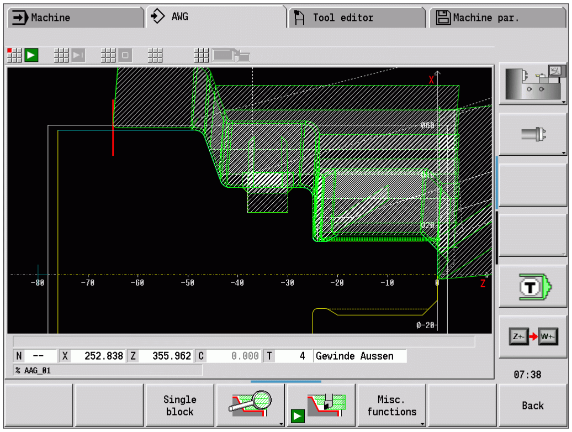
Generating and saving a working plan

Generating a working plan

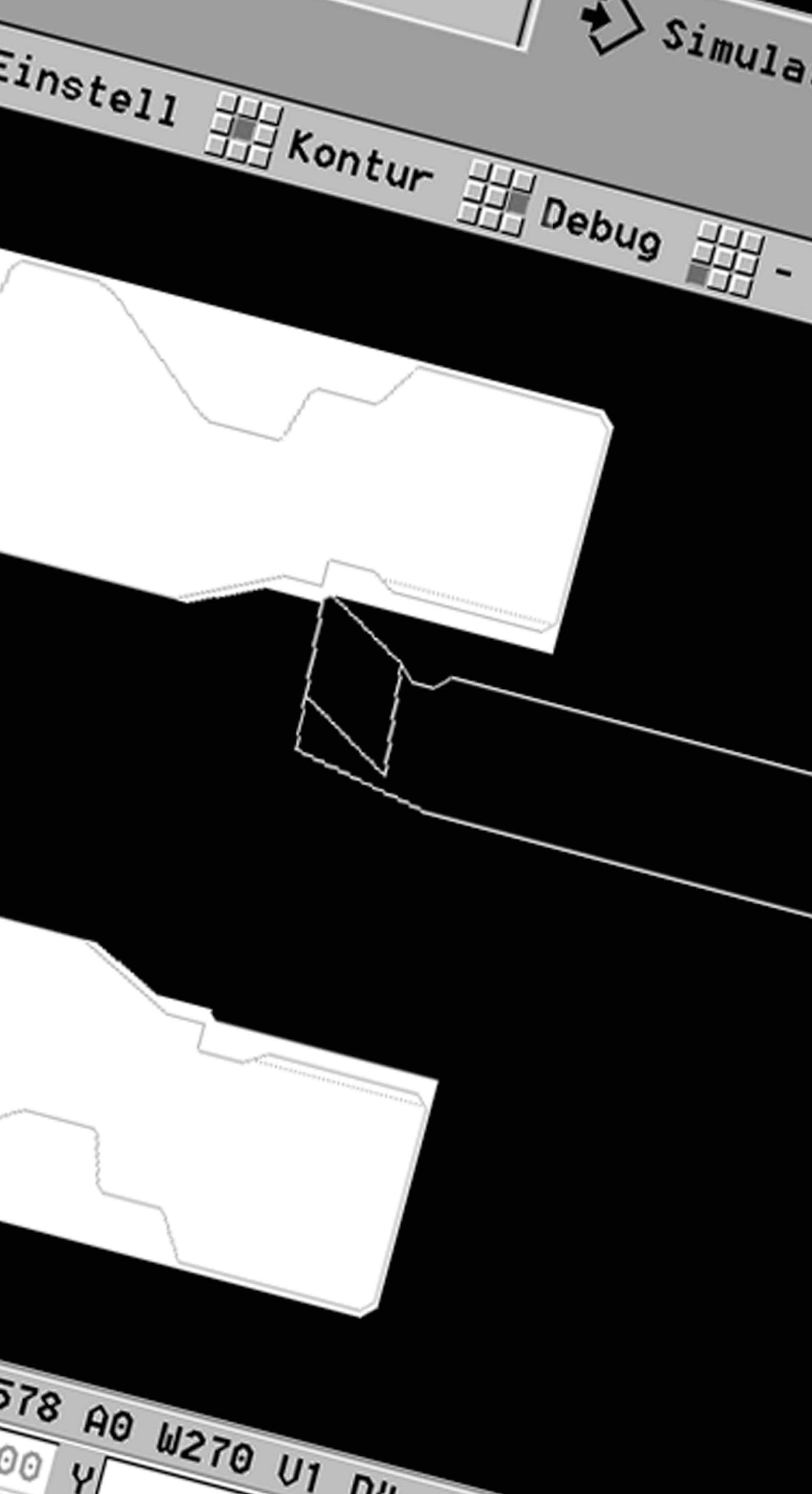
- Select "TURN PLUS > AWG."
- Start the AWG control graphic

Saving the program

- Press the "Back" soft key to return to the TURN PLUS menu.
- Press the "Back" soft key to return to the Program view
- Check/edit the file name and press the "Save" soft key
- TURN PLUS saves the NC program



> The AWG generates the work blocks according to the machining sequence and the settings of the machining parameters.



8

B axis

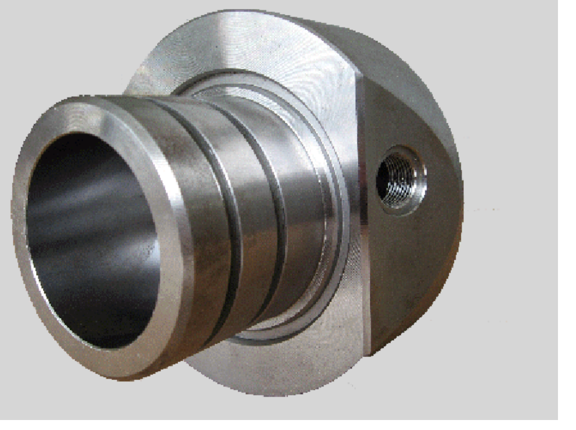
8.1 Fundamentals

Tilted working plane

The machine tool builder determines the scope of function and behavior of the B axis. The machine manual provides further information.

Tilted working plane

The B axis makes it possible to drill, bore and mill in oblique planes. To make programming easy, the coordinate system is tilted in such a way that you can define the drilling patterns and milling contours in the YZ plane. The actual drilling or milling operation is then performed in the tilted plane (see "Tilting the working plane G16" on page 498).



The separation of contour description and machining also applies to machining operations in tilted planes. Contour regeneration is not available.

Contours in tilted planes are identified with the section code SURFACE_Y (see "LATERAL_Y section" on page 50).

The control supports NC program creation with the B axis in DIN PLUS and smart.Turn.

The **graphical simulation** shows the machining operation in a tilted working plane in the familiar lathe and front windows, as well as in the "side view (YZ)."

If you are using a tool with an angled tool holder you can also use the tilted working plane without the B axis. Define the angle for the tool holder as angular offset **RW** in the tool description.

Tools for the B axis

Another advantage of the B axis is that it allows flexible use of the tools during turning operations. By tilting the B axis and rotating the tool you can bring it into positions that enable you to use one and the same tool to machine in the longitudinal and transverse (or radial and axial) directions on the main and opposing spindles.

In this way, you need fewer tools and fewer tool changes.

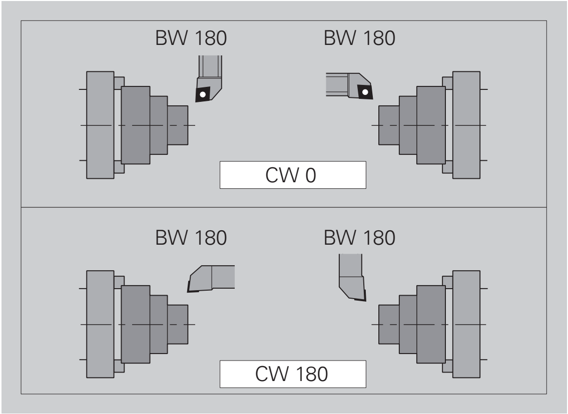
Tool data: All tools are described in the tool database by specifying the X, Z and Y dimensions as well as the compensation values. These dimensions are referenced to the **tilt angle B=0°** (reference position).

Another parameter that is maintained in the tool database is the **position angle** CW. It defines the working positions of tools that are not driven tools (turning tools).

The tilt angle of the B axis is not maintained with the tool data. This angle needs to be defined in the tool call or when inserting the tool.

Tool orientation and position display: For turning tools, the position the tool tip is calculated based on the orientation of the cutting edge.

The control calculates the tool orientation of lathe tools by means of the tool angle and point angle.

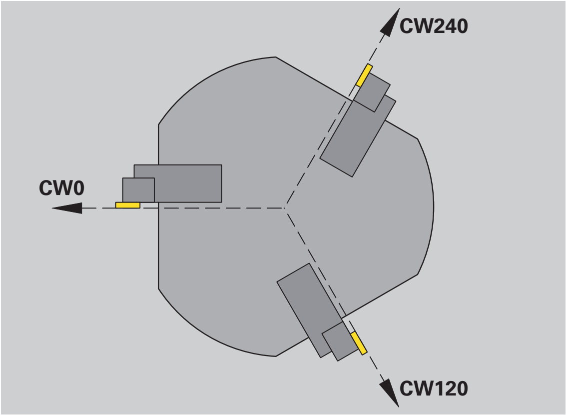


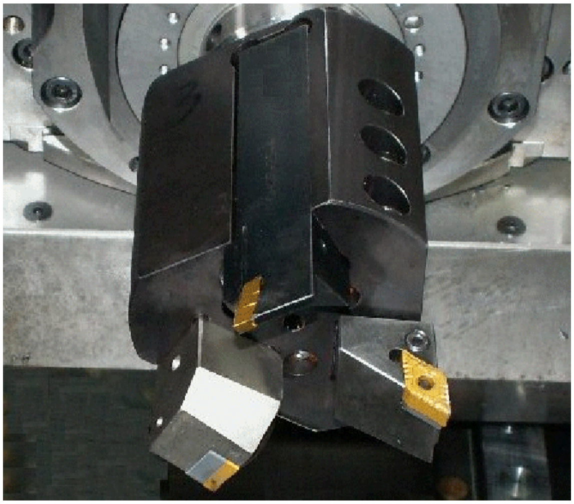
Multipoint tools for the B axis

If several tools are mounted on a tool holder, this is referred to as a "multipoint tool." Each cutting edge (tool) of a multipoint tool is assigned a separate ID number and description.

The **position angle**, which is identified by "CW" in the figure, is included in the tool data. When a cutting edge (tool) of a multipoint tool is activated, the CNC PILOT will rotate the multipoint tool into the correct position. The position is determined from the position angle, to which the offset position angle from the tool change routine is added. This allows inserting the tool either in the "normal" attitude or "upside down."

The photo shows a multipoint tool with three cutting edges.





8.2 Compensation with the B axis

Compensation during program run

Tool compensation: Enter the compensation values determined in the tool compensation form. Also define further functions that were active while machining the measured surface:

- Tilt angle of the B axis **`BW`**
- Position angle of the tool **`CW`**
- Kinematics **`KM`**
- Plane **`G16`**

The control converts the measured data into dimensions referenced to the position B=0 and saves them in the tool database.

- Select the **`Tool/Add correct.`** soft key during program run.
- The control opens "Set the tool compensation" in the dialog box.
- Enter new values.
- Press the **`Save`** soft key.

In the "T" box (machine display), the control indicates the compensation values referenced to the current B axis angle and the tool position angle.

- The control saves the tool compensation data in the tool database, together with the other tool data.
- If the B axis is tilted, the control takes the tool compensation data into account when calculating the tool tip position.

Additive compensation values are independent of the tool data. The compensation values are effective in the X, Y and Z directions. Tilting the B axis has no influence on additive compensation values.

8.3 Simulation

Simulation of the tilted plane

Contour graphics: The simulation displays the YZ view of the workpiece and the contours of the tilted planes in the **side view**. To represent the drilling patterns and milling contours perpendicularly to the tilted plane, i.e. without distortion, the simulation ignores the rotation of the coordinate system and a shift within the rotated coordinate system.

With contour graphics for tilted planes, please note the following:

- The parameter "K" of G16 or SURFACE_Y defines the "start" of the drilling pattern or milling contour in the Z direction.
- The drilling patterns and milling contours are drawn perpendicularly to the tilted plane. This results in a "shift" relative to the turning contour.

Milling, drilling and boring operations: When you use the **side view** to display the tool paths in the tilted plane, the same rules apply as for the contour graphics.

When working in tilted planes, the **front window** shows the "outline" of the tool. The tool width is simulated true to scale. In this way, you can check the overlap of milling paths. The tool paths are also represented true to scale (in perspective view) as line graphics.

In all "additional windows," the simulation shows the tool and the cutting path when the tool is perpendicular to the relevant plane. A tolerance of +/– 5° is taken into account. When the tool is not perpendicular to the plane, it is represented as a "light dot" and the tool path is depicted as a line.

Example: "Contour in tilted plane"

```
...
FINISHED PART
N2 G0 X0 Z0
N3 G1 X50
N4 G1 Z–50
N5 G1 X0
N6 G1 Z0
MANTEL_Y X50 C0 B80 I25 K-10 H0
[SURFACE_Y]
N7 G386 Z0 Ki10 B–30 X50 C0  [Single
surface]
MANTEL_Y X50 C0 B20 I25 K-20 H1
[SURFACE_Y]
N8 G384 Z–10 Y10 X50 R10 P5  [Full circle]
...
```

Displaying the coordinate system

The simulation can show the shifted/rotated coordinate system in the "lathe window," if required. To use this feature, you need to stop the simulation.

-/+ ▶ Press the Plus/Minus key, The simulation displays the current coordinate system.

The coordinate system disappears when the next command is simulated or when you press the Plus/Minus key once again.

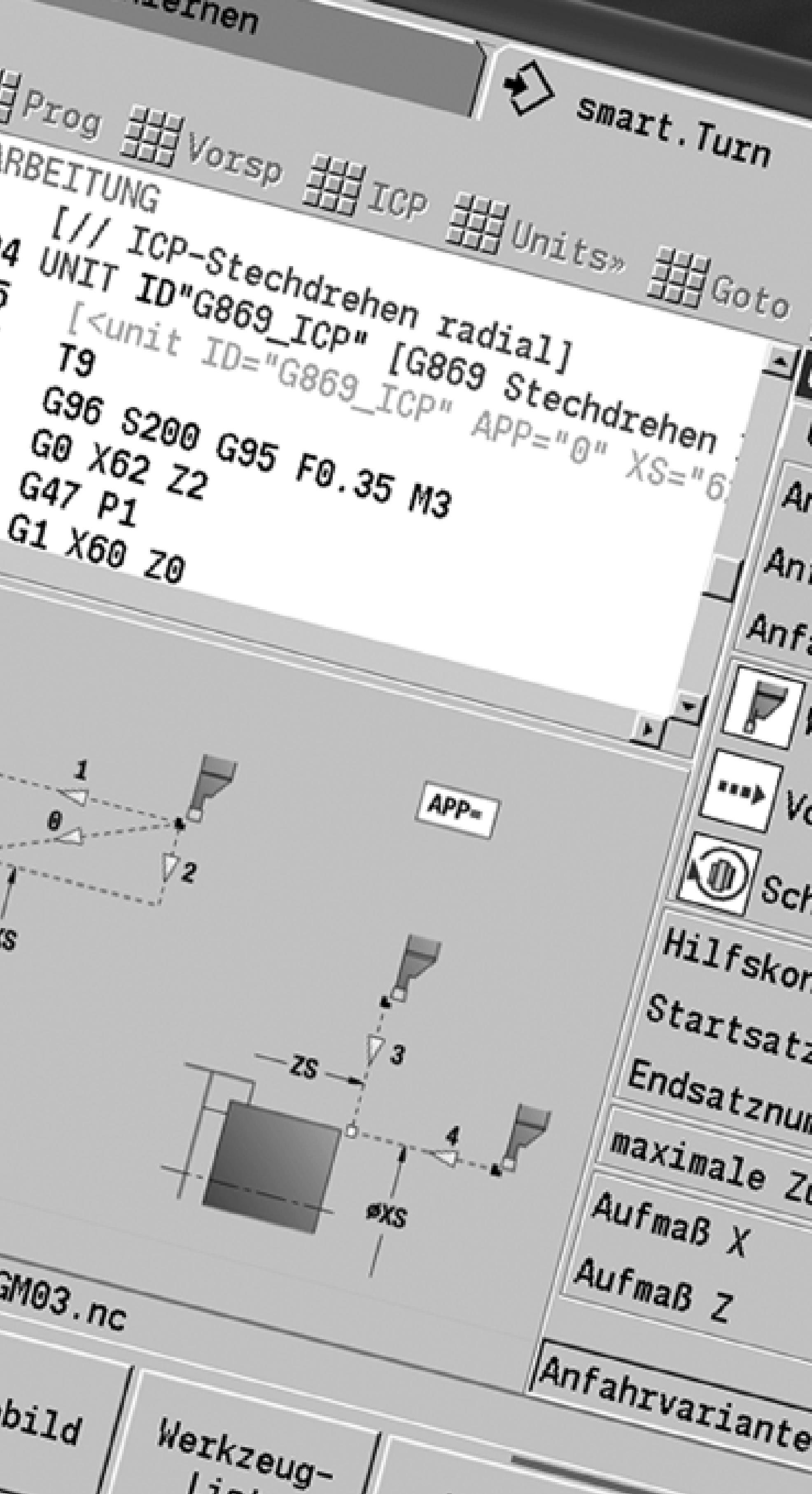
Position display with the B and Y axes

The following boxes of the display cannot be edited:

- **N:** Block number of the NC source block
- **X, Z, C:** Position values (actual values)

The other boxes can be set with the Split-Screen Layout key (three arrows arranged in a circle):

- Default settings (values of the selected slide):
  - **Y:** Position value (actual value)
  - **T:** Tool data with turret pocket, (in "(..)") and ID number
- B axis settings:
  - **B:** Tilt angle of the B axis
  - **G16/B:** Angle of the tilted plane



9

Overview of units

9.1 Units—"Turning" group

"Roughing" group

Unit	Description	Page
G810_ICP	**G810 Longitudinal in ICP** Roughing an ICP contour longitudinally	Page 65
G820_ICP	**G820 Transverse in ICP** Roughing an ICP contour transversely	Page 66
G830_ICP	**G830 Contour parallel in ICP** Roughing parallel to the contour in ICP	Page 67
G835_ICP	**G835 Bidirectional in ICP** Roughing an ICP contour in two directions	Page 68
G810_G80	**G810 Longitudinal, direct** Longitudinal roughing with direct contour input	Page 69
G820_G80	**G820 Transverse, direct** Transverse roughing with direct contour input	Page 70

"Finishing" group

Unit	Description	Page
G890_ICP	**G890 Contouring in ICP** Finishing an ICP contour	Page 115
G890_G80_L	**G890 Contouring, direct longitdnl.** Longitudinal finishing with direct contour input	Page 117
G890_G80_P	**G890 Contouring, direct transverse** Transverse finishing with direct contour input	Page 118
G85x_DIN_E_F_G	**G890 Relief, type E, F, DIN76** Finishing the undercuts according to DIN509 type E and F and the thread undercut DIN76	Page 119

"Recessing" group

Unit	Description	Page
G860_ICP	**G860 Contour recess in ICP** Recessing an ICP contour	Page 71
G869_ICP	**G869 Recess turning in ICP** Recess turning an ICP contour	Page 72
G860_G80	**G860 Contour recess, direct** Contour recessing with direct contour input	Page 73
G869_G80	**G869 Recess turning, direct** Recess turning with direct contour input	Page 74
G859_Cut_off	**G859 Parting** Parting a bar with direct position input	Page 75
G85x_Cut_H_K_U	**G85X Undercutting (H, K, U)** Make undercuts of shape H, K and U	Page 76

"Thread" group

Unit	Description	Page
G32_MAN	**G32 Thread, simple** Thread with direct contour definition	Page 123
G31_ICP	**G31 Thread, ICP** Thread on any desired ICP contour	Page 124
G352_API	**G352 API thread** API thread with direct contour definition	Page 126
G32_KEG	**G32 Tapered thread** Tapered thread with direct contour definition	Page 127

9.2 Units—"Drilling" group

"Centric drilling" group

Unit	Description	Page
G74_Zentr	**G74 Centric drilling** Drilling and pecking with X=0	Page 78
G73_Zentr	**G73 Centric tapping** Tapping with X=0	Page 80

"ICP drilling, C axis" group

Unit	Description	Page
G74_ICP_C	**G74 ICP drilling, C axis** Drilling and pecking with ICP pattern	Page 100
G73_ICP_C	**G73 ICP tapping, C axis** Tapping with ICP pattern	Page 101
G72_ICP_C	**G72 ICP boring, countersinking in C axis** Tapping with ICP pattern	Page 102

"C-axis face drilling" group

Unit	Description	Page
G74_Bohr_Stirn_C	**G74 Single hole** Drilling and pecking a single hole	Page 82
G74_Lin_Stirn_C	**G74 Linear pattern drilling** Drilling and pecking a linear hole pattern	Page 84
G74_Cir_Stirn_C	**G74 Circ. pattern drilling** Drilling and pecking a circular hole pattern	Page 86
G73_Gew_Stirn_C	**G73 Tapping** Tapping a single hole	Page 88
G73_Lin_Stirn_C	**G73 Thread, linear pattern** Tapping a linear hole pattern	Page 89
G73_Cir_Stirn_C	**G73 Thread, circular pattern** Tapping a circular hole pattern	Page 90

"C-axis lateral surface drilling" group

Unit	Description	Page
G74_Bohr_Mant_C	**G74 Single hole** Drilling and pecking a single hole	Page 91
G74_Lin_Mant_C	**G74 Linear pattern drilling** Drilling and pecking a linear hole pattern	Page 93
G74_Cir_Mant_C	**G74 Circ. pattern drilling** Drilling and pecking a circular hole pattern	Page 95
G73_Gew_Mant_C	**G73 Tapping** Tapping a single hole	Page 97
G73_Lin_Mant_C	**G73 Thread, linear pattern** Tapping a linear hole pattern	Page 98
G73_Cir_Mant_C	**G73 Thread, circular pattern** Tapping a circular hole pattern	Page 99

9.3 Units—"Predrilling in C axis" group

"Predrilling in C-axis, face" group

Unit	Description	Page
DRILL_STI_KON_C	**G840 Predrill face, contour milling, figures** Determine the predrilling position and machine a hole	Page 103
DRILL_STI_840_C	**G840 Predrill face, ICP contour milling** Determine the predrilling position and machine a hole	Page 105
DRILL_STI_TASC	**G845 Predrill face, pocket milling, figures** Determine the predrilling position and machine a hole	Page 106
DRILL_STI_845_C	**G845 Predrill face, ICP pocket milling** Determine the predrilling position and machine a hole	Page 108

"Predrilling in C-axis, lateral surface" group

Unit	Description	Page
DRILL_MAN_KON_C	**G840 Predrill latrl., contour milling, figures** Determine the predrilling position and machine a hole	Page 109
DRILL_MAN_840_C	**G840 Predrill lateral surf., ICP contour milling** Determine the predrilling position and machine a hole	Page 111
DRILL_MAN_TAS_C	**G845 Predrill lateral surf., pocket milling, figures** Determine the predrilling position and machine a hole	Page 112
DRILL_MAN_845_C	**G845 Predrill lateral surf., ICP pocket milling** Determine the predrilling position and machine a hole	Page 114

9.4 Units—"Milling in C axis" group

"Milling in C-axis, face" group

Unit	Description	Page
G791_Nut_Stirn_C	**G791 Linear slot** Milling a linear slot	Page 129
G791_Lin_Stirn_C	**G791 Linear slot pattern** Milling of linear slots in a linear pattern	Page 130
G791_Cir_Stirn_C	**G791 Circular slot pattern** Milling of linear slots in a circular pattern	Page 131
G797_STIRNFR_C	**G797 Face milling** Milling various figures as islands	Page 132
G799_GewindeFR_C	**G799 Thread milling** Inside thread milling in a single hole	Page 133
G840_FIG_STIRN_C	**G840 Contour milling, figures** Milling figures inside, outside and on the contour	Page 134
G84X_FIG_STIRN_C	**G84x Pocket milling, figures** Roughing out closed figures, inside	Page 137
G801_GRA_STIRN_C	**G801 Engraving** Engraving characters strings on the face	Page 140

"Milling in C axis, face" group

Unit	Description	Page
G840_Kon_C_STIRN	**G840 Contour milling, ICP** Machining ICP contours on the face inside, outside and on the contour	Page 136
G845_TAS_C_STIRN	**G845 Pocket milling, ICP** Inside rough-out of closed ICP contours on the face	Page 139
G840_ENT_C_STIRN	**G840 Deburring** Deburring ICP contours on the face	Page 141

"C-axis lateral surface milling" group

Unit	Description	Page
G792_NUT_MANT_C	**G792 Linear slot** Milling a linear slot	Page 142
G792_LIN_MANT_C	**G792 Linear slot pattern** Milling of linear slots in a linear pattern	Page 143
G792_CIR_MANT_C	**G792 Circular slot pattern** Milling of linear slots in a circular pattern	Page 144
G798_Wendelnut_C	**G798 Helical slot milling** Milling a thread-shaped helical slot	Page 145
G840_FIG_MANT_C	**G840 Contour milling, figures** Milling figures inside, outside and on the contour	Page 146
G84x_FIG_MANT_C	**G84x Pocket milling, figures** Roughing out closed figures, inside	Page 149
G802_GRA_MANT_C	**G802 Engraving** Engraving characters strings on the lateral surface	Page 152

"C-axis lateral surface milling" group

Unit	Description	Page
G840_Kon_C_Mant	**G840 Contour milling, ICP** Machining ICP contours on the lateral surface inside, outside and on the contour	Page 148
G845_TAS_C_MANT	**G845 Pocket milling, ICP** Inside rough-out of closed ICP contours on the lateral surface	Page 151
G840_ENT_C_MANT	**G840 Deburring** Deburring ICP contours on the lateral surface	Page 153

9.5 Units — "Drilling, predrilling in Y axis" group

"ICP drilling, Y axis" group

Unit	Description	Page
G74_ICP_Y	**G74 ICP drilling, Y axis** Drilling and pecking with ICP pattern	Page 162
G73_ICP_Y	**G73 ICP tapping, Y axis** Tapping with ICP pattern	Page 163
G72_ICP_Y	**G72 ICP boring, countersinking in Y axis** Tapping with ICP pattern	Page 164

"Predrilling in Y axis" group

Unit	Description	Page
DRILL_STI_840_Y	**G840 ICP predrilling, contour milling in XY plane** Determine the predrilling position and machine a hole	Page 165
DRILL_STI_845_Y	**G845 ICP predrilling, pocket milling in XY plane** Determine the predrilling position and machine a hole	Page 166
DRILL_MAN_840_Y	**G840 ICP predrilling, contour milling in YZ plane** Determine the predrilling position and machine a hole	Page 167
DRILL_MAN_845_Y	**G845 ICP predrilling, pocket milling in YZ plane** Determine the predrilling position and machine a hole	Page 168

9.6 Units—"Milling in Y axis" group

"Milling in front face" group (XY plane)

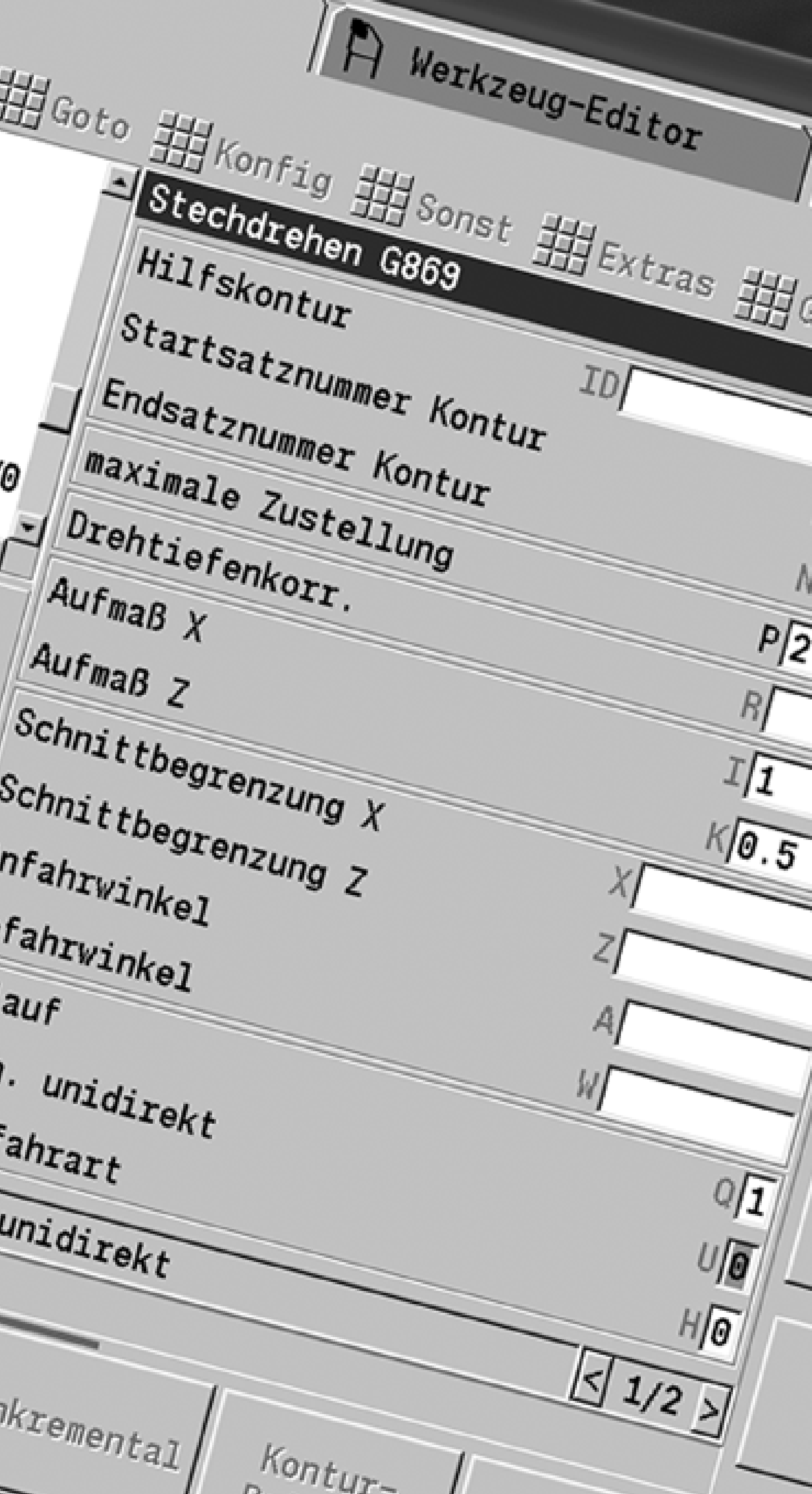
Unit	Description	Page
G840_Kon_Y_Stirn	**G840 Contour milling** Machining contours in the XY plane inside, outside and on the contour	Page 169
G845_Tas_Y_Stirn	**G845 Pocket milling** Inside rough-out of closed contours in the XY plane	Page 170
G840_ENT_Y_STIRN	**G840 Deburring** Deburring contours in the XY plane	Page 174
G801_GRA_STIRN_C	**G841 Single surface** Milling a single surface (flat) in the XY plane	Page 171
G840_Kon_C_STIRN	**G843 Centric polygon** Milling a centric polygon in the XY plane	Page 172
G803_GRA_Y_STIRN	**G803 Engraving** Engraving character strings in the XY plane	Page 173
G800_GEW_Y_STIRN	**G800 Thread milling** Milling a thread in an existing hole in the XY plane	Page 175

"Milling in lateral surface" group (YZ plane)

Unit	Description	Page
G840_Kon_Y_Mant	**G840 Contour milling** Machining contours in the YZ plane inside, outside and on the contour	Page 176
G845_Tas_Y_Mant	**G845 Pocket milling** Inside rough-out of closed contours in the YZ plane	Page 177
G840_ENT_Y_MANT	**G840 Deburring** Deburring contours in the YZ plane	Page 181
G801_GRA_STIRN_C	**G841 Single surface** Milling a single surface (flat) in the YZ plane	Page 178
G840_Kon_C_STIRN	**G843 Centric polygon** Milling a centric polygon in the YZ plane	Page 179
G804_GRA_Y_MANT	**G803 Engraving** Engraving character strings in the YZ plane	Page 180
G806_GEW_Y_MANT	**G800 Thread milling** Milling a thread in an existing hole in the YZ plane	Page 182

9.7 Units—"Special units" group

Unit	Description	Page
START	**Program beginning (START)** For functions required at the beginning of the program	Page 154
C_AXIS_ON	**C axis on** Activate C-axis interpolation	Page 156
C_AXIS_OFF	**C axis off** Deactivate C-axis interpolation	Page 156
SUBPROG	**Subprogram call** Calling any desired subprogram	Page 157
REPEAT	**Process logic—repetition** Describing a WHILE loop to repeat parts of the program	Page 158
END	**Program end (END)** For functions required at the end of the program	Page 159



10

Overview of G functions

10.1 Section codes

Program section codes			Program section codes	
Program head			**Y-axis contours**	
PROGRAMMKOPF / HEADER	Page 47		STIRN_Y / FACE_Y	Page 49
REVOLVER / TURRET	Page 48		RUECKSEITE_Y / REAR_Y	Page 49
SPANNMITTEL / CHUCKING EQUIPMENT	Page 48		MANTEL_Y / LATERAL_Y	Page 50
Contour definition			**Workpiece machining**	
ROHTEIL / BLANK	Page 49		BEARBEITUNG / MACHINING	Page 51
HILFSROHTEIL / AUXIL_BLANK	Page 49		ENDE / END	Page 51
FERTIGTEIL / FINISHED PART	Page 49		**Subroutines**	
HILFSKONTUR / AUXIL_CONTOUR	Page 49		UNTERPROGRAMM / SUBPROGRAM	Page 51
C-axis contours			RETURN	Page 51
STIRN / FACE_C	Page 49		**Others**	
RUECKSEITE / REAR_C	Page 49		CONST	Page 52
MANTEL / LATERAL_C	Page 49		VAR	Page 52

10.2 Overview of G commands in the CONTOUR section

G commands for turning contours

Turning contour				Turning contour		
Workpiece-blank definition				**Contour form elements**		
G20-Geo	Chuck part, cylinder/tube	Page 194		G22-Geo	Recess (standard)	Page 200
G21-Geo	Cast part	Page 194		G23-Geo	Recess/relief turn	Page 202
Basic contour elements				G24-Geo	Thread with undercut	Page 204
G0-Geo	Starting point of contour	Page 195		G25-Geo	Undercut contour	Page 205
G1-Geo	Line segment	Page 196		G34-Geo	Thread (standard)	Page 209
G2-Geo	Circular arc cw with incremental center dimensioning	Page 198		G37-Geo	Thread (general)	Page 210
G3-Geo	Circular arc ccw with incremental center dimensioning	Page 198		G49-Geo	Bore hole at turning center	Page 212
G12-Geo	Circular arc cw with absolute center dimensioning	Page 199		**Help commands for contour definition**		
G13-Geo	Circular arc ccw with absolute center dimensioning	Page 199		Overview: Attributes for contour description		Page 213
				G38-Geo	Feed rate reduction	Page 213
				G44	Separation point	Page 215
				G52-Geo	Oversize	Page 215
				G95-Geo	Feed per revolution	Page 216
				G149-Geo	Additive compensation	Page 216

G commands for C-axis contours

C-axis contour				C-axis contour		
Overlapping contours				**Overlapping contours**		
G308-Geo	Beginning of pocket/island	Page 217		G309-Geo	End of pocket/island	Page 217
Front and rear face contours				**Lateral surface contours**		
G100-Geo	Starting point of contour, face	Page 223		G110-Geo	Starting point of lateral surface contour	Page 232
G101-Geo	Line segment, face	Page 224		G111-Geo	Line segment, lateral surface	Page 233
G102-Geo	Arc cw, face	Page 225		G112-Geo	Arc cw, lateral surface	Page 234
G103-Geo	Arc ccw, face	Page 225		G113-Geo	Arc ccw, lateral surface	Page 234
G300-Geo	Bore hole, face	Page 226		G310-Geo	Bore hole on lateral surface	Page 235
G301-Geo	Linear slot, face	Page 227		G311-Geo	Linear slot on lateral surface	Page 236
G302-Geo	Circular slot cw, face	Page 227		G312-Geo	Circular slot cw, lateral surface	Page 236
G303-Geo	Circular slot ccw, face	Page 227		G313-Geo	Circular slot ccw, lateral surface	Page 236
G304-Geo	Full circle, face	Page 228		G314-Geo	Full circle, lateral surface	Page 237
G305-Geo	Rectangle, face	Page 228		G315-Geo	Rectangle, lateral surface	Page 237
G307-Geo	Polygon, face	Page 229		G317-Geo	Polygon, lateral surface	Page 238
G401-Geo	Linear pattern, face	Page 230		G411-Geo	Linear pattern, lateral surface	Page 239
G402-Geo	Circular pattern, face	Page 231		G412-Geo	Circular pattern, lateral surface	Page 240

G commands for Y-axis contours

Y-axis contour				Y-axis contour		
XY plane				**YZ plane**		
G170-Geo	Starting point of contour in XY plane	Page 480		G180-Geo	Starting point of contour in YZ plane	Page 489
G171-Geo	Line segment in XY plane	Page 480		G181-Geo	Line segment in YZ plane	Page 489
G172-Geo	Arc cw in XY plane	Page 481		G182-Geo	Arc cw in YZ plane	Page 490
G173-Geo	Arc ccw in XY plane	Page 481		G183-Geo	Arc ccw in YZ plane	Page 490
G370-Geo	Hole in XY plane	Page 482		G380-Geo	Hole in YZ plane	Page 491
G371-Geo	Linear slot in XY plane	Page 483		G381-Geo	Linear slot in YZ plane	Page 491
G372-Geo	Circular slot cw in XY plane	Page 484		G382-Geo	Circular slot cw in YZ plane	Page 492
G373-Geo	Circular slot ccw in XY plane	Page 484		G383-Geo	Circular slot ccw in YZ plane	Page 492
G374-Geo	Full circle in XY plane	Page 484		G384-Geo	Full circle in YZ plane	Page 492
G375-Geo	Rectangle in XY plane	Page 485		G385-Geo	Rectangle in YZ plane	Page 493
G377-Geo	Polygon in XY plane	Page 485		G387-Geo	Polygon in YZ plane	Page 493
G471-Geo	Linear pattern in XY plane	Page 486		G481-Geo	Linear pattern in YZ plane	Page 494
G472-Geo	Circular pattern in XY plane	Page 487		G482-Geo	Circular pattern in YZ plane	Page 495
G376-Geo	Single surface in XY plane	Page 488		G386-Geo	Single surface in XY plane	Page 496
G477-Geo	Centric polygon in XY plane	Page 488		G487-Geo	Centric polygon in XY plane	Page 496

10.3 Overview of G commands in the MACHINING section

G commands for turning

Turning—Basic functions		
Tool positioning without machining		
G0	Positioning at rapid traverse	Page 241
G14	Move to the tool change position	Page 242
G140	Define the tool change position	Page 242
G701	Rapid traverse to machine coordinates	Page 241
Simple linear and circular movements		
G1	Linear movement	Page 243
G2	Circular movement cw with incremental center dimensioning	Page 244
G3	Circular movement ccw with incremental center dimensioning	Page 244
G12	Circular movement cw with absolute center dimensioning	Page 245
G13	Circular movement ccw with absolute center dimensioning	Page 245
Feed rate and spindle speed		
Gx26	Speed limit *	Page 246
G64	Interrupted feed rate	Page 246
Gx93	Feed per tooth *	Page 247
G94	Feed per minute	Page 247
Gx95	Feed per revolution	Page 247
Gx96	Constant surface speed	Page 248
Gx97	Shaft speed	Page 248
Oversizes		
G50	Switch off oversize	Page 255
G52	Switch off oversize	Page 255
G57	Paraxial oversize	Page 255
G58	Contour-parallel oversize	Page 256

Turning—Basic functions		
Zero point shifts		
Overview: Zero point shifts		Page 251
G51	Zero point shift	Page 252
G56	Additive zero-point shift	Page 253
G59	Absolute zero point shift	Page 254
G152	Zero point shift, C axis	Page 329
G920	Deactivate zero point shifts	Page 374
G921	Deactivate zero point shift, tool dimensions	Page 374
G980	Activate zero point shift	Page 377
G981	Activate zero point shift, tool dimensions	Page 377
Safety clearances		
G47	Set safety clearances	Page 257
G147	Safety clearance (milling)	Page 257
Tool-tip radius compensation (TRC/MCRC)		
G40	Switch off TRC/MCRC	Page 249
G41	TRC/MCRC, left	Page 250
G42	TRC/MCRC, right	Page 250
Tools, types of compensation		
T	Insert the tool	Page 258
G148	(Changing the) cutter compensation	Page 259
G149	Additive compensation	Page 260
G150	Compensate right tool tip	Page 261
G151	Compensate left tool tip	Page 261

Cycles for turning

Turning—Cycles				Turning—Cycles		
Simple turning cycles				**Contour-based turning cycles**		
G80	Cycle end / simple contours	Page 286		G740	Contour repeat cycle	Page 277
G81	Simple longitudinal roughing	Page 411		G741	Contour repeat cycle	Page 277
G82	Simple face roughing	Page 412		G810	Longitudinal roughing cycle	Page 264
G83	Contour repeat cycle	Page 413		G820	Face roughing cycle	Page 267
G86	Simple recessing cycle	Page 414		G830	Contour-parallel roughing cycle	Page 270
G87	Transition radii	Page 415		G835	Contour-parallel with neutral tool	Page 273
G88	Chamfer	Page 415		G860	Universal recessing cycle	Page 275
Drilling cycles				G869	Recess turning cycle	Page 278
G36	Tapping	Page 320		G870	Simple recessing cycle G22	Page 281
G71	Simple drilling cycle	Page 315		G890	Finishing cycle	Page 282
G72	Boring, countersinking, etc.	Page 317		**Thread cycles**		
G73	Tapping cycle	Page 318		G31	Thread cycle	Page 293
G74	Deep-hole drilling cycle	Page 321		G32	Single thread cycle	Page 297
Undercuts				G33	Single thread cut (Thread single path)	Page 299
G25	Undercut contour	Page 205		G35	Metric ISO thread	Page 301
G85	Undercut	Page 306		G350	Simple longitudinal thread	
G851	Undercut DIN 509 E, direct	Page 308		G351	Simple longitudinal multi-start thread	
G852	Undercut DIN 509 F, direct	Page 309		G352	Tapered API thread	Page 302
G853	Undercut DIN 76 F thread, direct	Page 310		G36	Tapping	Page 320
G856	Undercut type U, direct	Page 311		G38	Metric ISO thread	Page 304
G857	Undercut type H, direct	Page 312		**Parting**		
G858	Undercut type K, direct	Page 313		G859	Parting cycle	Page 305

C-axis machining

C-axis machining				C-axis machining		
C axis						
G120	Reference diameter, lateral-surface machining	Page 329				
G152	Zero point shift, C axis	Page 329				
G153	Standardize C axis	Page 330				
Single path—Front/rear face machining				**Single path—Lateral-surface machining**		
G100	Rapid traverse, face	Page 331		G110	Rapid traverse, lateral surface	Page 335
G101	Linear path, face	Page 332		G111	Linear path, lateral surface	Page 336
G102	Circular path cw, face	Page 333		G112	Circular path cw, lateral surface	Page 337
G103	Circular path ccw, face	Page 333		G113	Circular path ccw, lateral surface	Page 337
Figures—Front/rear face machining				**Figures—Lateral-surface machining**		
G301	Linear slot, face	Page 287		G311	Linear slot on lateral surface	Page 289
G302	Circular slot cw, face	Page 287		G312	Circular slot cw, lateral surface	Page 290
G303	Circular slot ccw, face	Page 287		G313	Circular slot ccw, lateral surface	Page 290
G304	Full circle, face	Page 288		G314	Full circle, lateral surface	Page 290
G305	Rectangle, face	Page 288		G315	Rectangle, lateral surface	Page 291
G307	Polygon, face	Page 288		G317	Polygon, lateral surface	Page 291
Milling cycles, face				**Milling cycles, lateral surface**		
G791	Linear slot, face	Page 339		G792	Linear slot, lateral surface	Page 340
G793	Contour milling, direct	Page 341		G794	Contour milling, direct	Page 343
G797	Area milling (face milling)	Page 345		G798	Helical slot milling	Page 347
G799	Thread milling					
Predrilling cycles				**Contour and pocket milling cycles**		
G840	Predrilling, contour milling	Page 349		G840	Contour milling	Page 351
G845	Predrilling, pocket milling	Page 359		G840	Deburring	Page 355
Engraving cycles				G845	Pocket milling	Page 360
G801	Engraving, face	Page 368		G846	Pocket milling, finishing	Page 364
G802	Engraving, lateral surface	Page 369		**Engraving cycles**		
Patterns				G801	Engraving, face	Page 368
G743	Pattern linear, face			G802	Engraving, lateral surface	Page 369
G745	Pattern circular, face				Character set for engraving	Page 366
G744	Linear pattern, lateral surface					
G746	Circular pattern, lateral surface					

Y-axis machining

Y-axis machining				Y-axis machining		
Working planes				**Milling cycles**		
G17	XY plane	Page 497		G841	Area milling, roughing	Page 504
G18	XZ plane (turning view)	Page 497		G842	Area milling, finishing	Page 505
G19	YZ plane	Page 497		G843	Centric polygon milling, roughing	Page 506
Tool positioning without machining				G844	Centric polygon milling, finishing	Page 507
G0	Positioning at rapid traverse	Page 499		G845	Predrilling, pocket milling	Page 509
G14	Move to the tool change position	Page 499		G845	Pocket milling, roughing	Page 510
G701	Rapid traverse to machine coordinates	Page 500		G846	Pocket milling, finishing	Page 514
Simple linear and circular movements				G800	Thread milling in XY plane	Page 518
G1	Linear movement	Page 501		G806	Thread milling in YZ plane	Page 519
G2	Circular movement cw with incremental center dimensioning	Page 502		G808	Hobbing	Page 520
G3	Circular movement ccw with incremental center dimensioning	Page 502		**Engraving cycles**		
G12	Circular movement cw with absolute center dimensioning	Page 503		G803	Engraving in XY plane	Page 516
G13	Circular movement ccw with absolute center dimensioning	Page 503		G804	Engraving in YZ plane	Page 517
					Character set for engraving	Page 366

Variable programming, program branches

Variable programming, program branches				Variable programming, program branches		
Programming with variables				**Data input and data output**		
# variables	Variable types	Page 390		INPUT	Input (# variables)	Page 387
PARA	Read configuration data	Page 396		WINDOW	Open output window (# variables)	Page 387
CONST	Constant definition	Page 399		PRINT	Output (# variables)	Page 388
VAR	Variable definition	Page 398		**Program branches, program repeats**		
Subroutines				IF..THEN..	Program branching	Page 400
Subprogram call		Page 404		WHILE..	Program repeat	Page 402
				SWITCH..	Program branching	Page 403

Other G functions

Other G functions				Other G functions		
G4	Dwell time	Page 371		G909	Interpreter stop	Page 373
G7	Precision stop ON	Page 371		G910	Switching measurement on/off	Page 473
G8	Precision stop OFF	Page 372		G911	Activate measuring path monitoring	Page 474
G9	Precision stop (blockwise)	Page 372		G912	Actual position capture	Page 474
G30	Converting and mirroring	Page 378		G913	End in-process measurement	Page 474
G44	Separation point	Page 215		G914	Deactivate measuring path monitoring	Page 474
G60	Deactivate protection zone	Page 372		G916	Traversing to a fixed stop	Page 382
G65	Display chucking equipment	Page 371		G919	Spindle override 100%	Page 373
G67	Load blank-part contour (graphics)	Page 371		G920	Deactivate zero point shift	Page 374
G99	Transformations of contours	Page 379		G921	Deactivate zero point shift, tool dimensions	Page 374
G702	Storing/loading contour follow-up	Page 370		G922	Tool end position	Page 374
G703	Contour follow-up ON/OFF	Page 370		G923	Handwheel offset in thread	Page 122
G720	Spindle synchronization	Page 380		G924	Fluctuating speed	Page 374
G901	Actual values in variables	Page 372		G925	Force reduction	Page 385
G902	Zero point shift in variables	Page 372		G927	Convert tool lengths	Page 375
G903	Lag error in variables	Page 372		G930	Sleeve monitoring	Page 386
G904	Read interpolator information	Page 373		G940	Automatically convert variables	Page 375
G905	C-angle offset	Page 381		G980	Activate zero point shift	Page 377
G908	Feed rate override 100%	Page 373		G981	Activate zero point shift, tool dimensions	Page 377

S

T

V

W

X

Y

Z

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 (8669) 31-0
FAX +49 (8669) 5061
e-mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
e-mail: service@heidenhain.de
Measuring systems ☎ +49 (8669) 31-3104
e-mail: service.ms-support@heidenhain.de
TNC support ☎ +49 (8669) 31-3101
e-mail: service.nc-support@heidenhain.de
NC programming ☎ +49 (8669) 31-3103
e-mail: service.nc-pgm@heidenhain.de
PLC programming ☎ +49 (8669) 31-3102
e-mail: service.plc@heidenhain.de
Lathe controls ☎ +49 (711) 952803-0
e-mail: service.hsf@heidenhain.de

www.heidenhain.de

685556-23 · Ver03 · SW05 · Printed in Germany · H · 8/2013