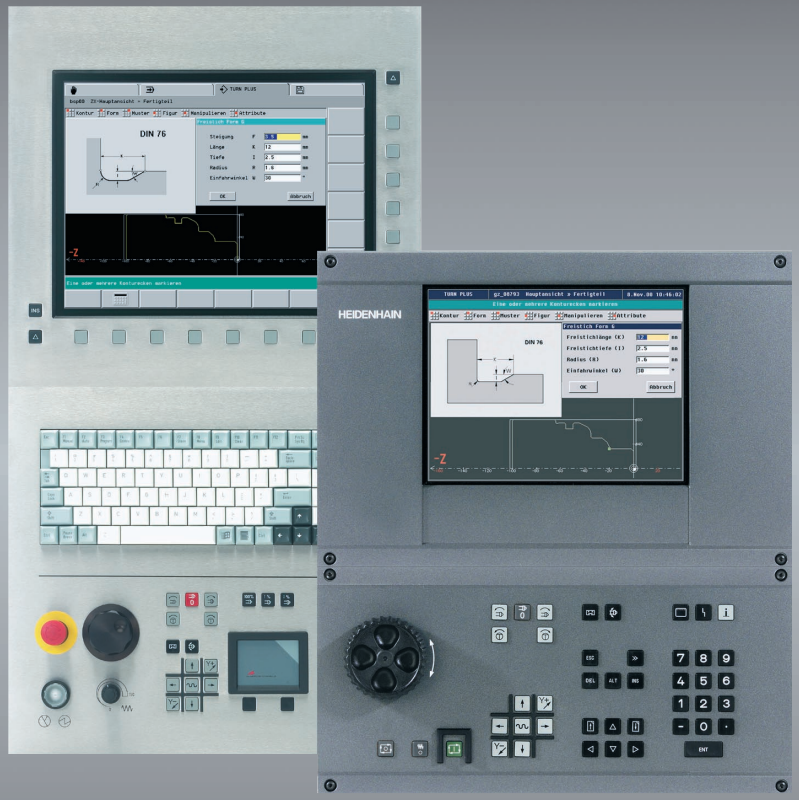


HEIDENHAIN

Pilot

CNC PILOT 4290

Software version 6.4/7.0

English (en)
6/2003

CNC PILOT 4290 V7.0 — Keyboard

Manual operating mode

Automatic operating mode

Programming modes (DIN PLUS, Simulation, TURN PLUS)

Organization modes (Parameter, Service, Transfer)

Display error status

Call info system

ESC (escape)

- Go back by one menu level
- Close dialog box, do not save data

"Continue key"

For special functions (e.g. marking)

DEL (delete)

- Deletes the list element
- Deletes the selected character or the character to the left of the cursor

ALT (alter)

- Edit the list element

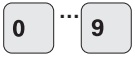
CNC PILOT 4290 V7.0 — Keyboard

INS (insert)

- Insert list element
- Close dialog box, save data

Numerals (0...9)

For entering numbers and selecting soft keys



Minus

For entering an algebraic sign

Decimal point

Enter

To confirm your input

Cursor keys

Moves the cursor by one position in the direction of the arrow (one character, one field, one line, etc.)

Page Up, Page Down

- Go to previous/next screen page
- Go to previous/next dialog box
- Switch between input windows

CNC PILOT 4290V6.4—Keyboard

Operating modes key
Call the selection of operating modes

Display error status

Call the info system

ESC
- Go back by one menu level
- Close dialog box, do not save data

>> (“continue” key)
For special functions (e.g. marking)

DEL
Delete key

ALT (alter)
- Edit the list element

INS (insert)
- Insert list element
- Close dialog box, save data

CNC PILOT 4290V6.4—Keyboard

Numerals (0...9)
For entering numbers and selecting soft keys

Minus
For entering an algebraic sign

Decimal point

Enter
To confirm your input

Cursor keys
Moves the cursor by one position in the direction of the arrow (one character, one field, one line, etc.)

Page Up, Page Down
- Go to previous/next screen page
- Go to previous/next dialog box
- Switch between input windows

The Pilot

... is your concise programming guide for the HEIDENHAIN CNC PILOT 4290 contouring control. For more comprehensive information on programming and operating, refer to the CNC PILOT User's Manual.

Certain symbols are used in the Pilot to denote specific types of information:

Important note!

Warning: Danger for the user or the machine!

Chapter in User's Manual. Here you will find more detailed information on the current topic.

The information in this Pilot applies to the CNC PILOT with the software number 340 340 460-xx (release 6.4) and the CNC PILOT with the software number 368 650-xx (release 7.0).

Contents

DIN Programming

NC blocks start with the letter "N" followed by a block number (with up to four digits).

Comments are enclosed in parentheses „[...]". They are located either at the end of an NC block or in a separate NC block.

Instructions for operation
During editing, the CNC PILOT shows programmed **contours** in a maximum of two **simulation windows**. You can select the windows from the DIN PLUS main menu (Menu item "Graphics—Windows").

- The starting point of the contour will be marked with a "small box"
- If the cursor is located on a block from "blank or finished part", the corresponding contour element will be indicated in red in the simulation window ("Contour display")

- Additions/changes to the contour will only be considered if the "Graphics" menu item is reactivated.
- Unambiguous NC block numbers are a prerequisite for the contour display!

- For programming variables, see "CNC PILOT 4290 User's Manual"
- For programming in the Y axis, see "CNC PILOT 4290 with Y Axis User's Manual"

Program section codes

When you create a new DIN program, certain program section codes are already entered. Delete or add codes, depending on the task. A DIN program must include the codes "MACHINING" and "END."

Overview of program section codes
PROGRAMMKOPF [PROGRAM HEAD]
TURRET
CLAMPING DEVICE
ROHTEIL [BLANK]
FERTIGTEIL [FINISHED PART]
FRONT END
REAR END
CYLINDER SURFACE
AUXILIARY CONTOUR
BEARBEITUNG [MACHINING]
ENDE [END]
SUBPROGRAM
RETURN

PROGRAMMKOPF [PROGRAM HEAD]
The PROGRAM HEAD comprises:

- **Organizational information** (does not influence program execution)
- **Setup information** (does not influence program execution)
- **SLIDE**: NC program is only executed for the indicated slide – No input: NC program is executed for every slide (input: "$1, $2, ...")
- **UNIT**: unit of measurement "metric/inches"—No input: the unit set in control parameter 1 is used

The "Unit" can be programmed only when a new program is being created (set under PROGRAM HEAD). It is not possible to post-edit this entry.

TURRET x
contains the assignment for the tool carrier x (x: 1..6). If the tool is described in the data bank, enter the T number and the ID number. Alternately, you can define the tool parameters in the NC program.

Tool data input:

- **Call** the tool input: INS key
- **T-number:** position in the tool carrier
- **ID (identification number):** reference to the tool database– No input: tool data **is not** included in the tool database.

Simple tool:

- Only suitable for simple traverse paths and turning cycles (G0...G3, G12, G13; G81...G88).
- There is **no** regeneration of the contour.
- Cutter radius compensation is carried out.
- Data are not stored in the tool database ("Simple tools" have no ID).

Continued ▶

Enhanced input: No limitations for use of the tool (data is transferred to the tool database during program conversion.)

If you do not program **TURRET**, the tools entered in the turret table will be used.

CLAMPING DEVICE x
Defines the type of clamping device X used on the spindle (x: 1..4).

If you do not program **CLAMPING DEVICE**, the machining simulation assumes there is no clamping device (see also G65).

Parameters
H: Clamping device number (reference for G65) – Range: 1 ≤ H ≤ 9
ID: Identification number of clamping device
X: Clamping diameter
Q: Chucking shape – defines the position of the clamping device reference point (see G65)

ROHTEIL [BLANK]
Program section for the definition of the blank.

FERTIGTEIL [FINISHED PART]
Program section for the contour definition of the finished part.

Additional program section codes within the finished part definition:
- **FRONT END Z.. :** Section "Front end contour" – "Z.." defines the position of the front contour.
- **REAR SIDE Z.. :** Section "Rear side contour" – "Z.." defines the position of the rear side contour.
- **LATERAL SURFACE X.. :** section "Lateral surface contour" – "X.."
- **AUXILIARY CONTOUR**: indicates further contour definitions

If you have several independent contour definitions, then repeated use of the program section codes (FRONT END, REAR END, etc.) is permitted.

BEARBEITUNG [MACHINING]
Program section for the machining of the workpiece. MACHINING **must** be included in your program.

ENDE [END]
Ends your NC program. The code END **must** be included in your program (replaces M30).

SUBPROGRAM "12345678"
If you define a subprogram within your NC program (within the same file), it is identified with SUBPROGRAM, followed by the name of the subprogram (max. 8 characters).

RETURN
Ends your NC subprogram.

Blank material for cylinder/pipe G20-Geo

G20 defines the contour of a cylinder/hollow cylinder.

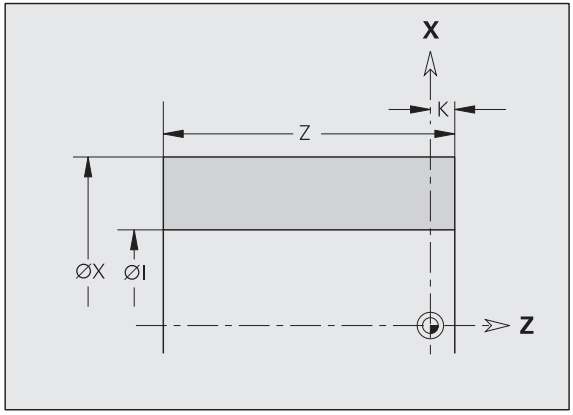
Parameters

X: ■ Diameter of cylinder/hollow cylinder
■ Diameter of circumference of polygonal blank

Z: Length of blank

K: Right edge (distance between workpiece datum and right edge)

I: Inside diameter for hollow cylinders



Cast part G21-Geo

G21 generates the contour of the blank part from the contour of the finished part – plus the "equidistant allowance P."

Parameters

P: Equidistant finishing allowance (reference: finished part contour)

Q: Bore holes yes/no – default: Q=0
■ Q=0: without bore holes
■ Q=1: with bore holes

Starting point of turning contour G0 Geo

G0 defines the starting point of a turning contour.

Parameters

X, Z: Starting point of the contour (X diameter value)

Line segment in a contour G1-Geo

G1 defines a line segment in a turning contour.

Parameters

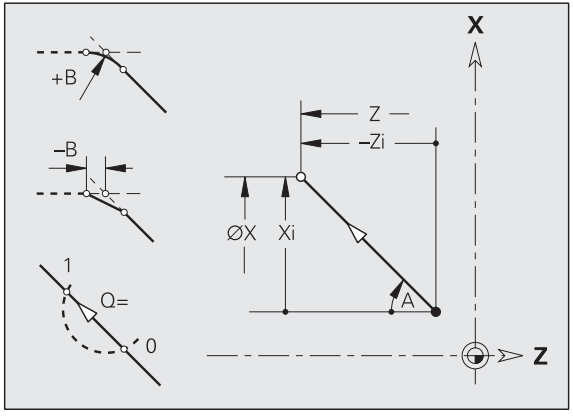
X, Z: End point (X diameter value)

A: Angle to rotary axis – for angle direction see illustration

Q: Select point of intersection – default: 0
- Q=0: Near intersection
- Q=1: Far intersection

B: Chamfer/rounding
- B is undefined: Tangential transition
- B=0: Nontangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E: Special feed-rate factor (0 < E ≤ 1) – default: 1
(special feed rate = active feed rate * E)



Circular arc in a contour

G2/G3-Geo – incremental, G12/G13-Geo – absolute center coordinates

G2/G3 or G12/G13 defined a circular arc in a contour. The direction of rotation is visible in the help graphic.

Parameters

X, Z: End point (X diameter value)

R: Radius

Q: Selection of intersection – default: 0
- Q=0: Far intersection
- Q=1: Near intersection

B: Chamfer/ rounding at end of circular arc
- B no entry: tangential transition
- B=0: no tangential transition
- B>0: Radius of rounding
- B<0: Width of chamfer

E: Special feed-rate factor (0 < E ≤ 1) – default: 1
(special feed rate = active feed rate * E)

With G2/G3:

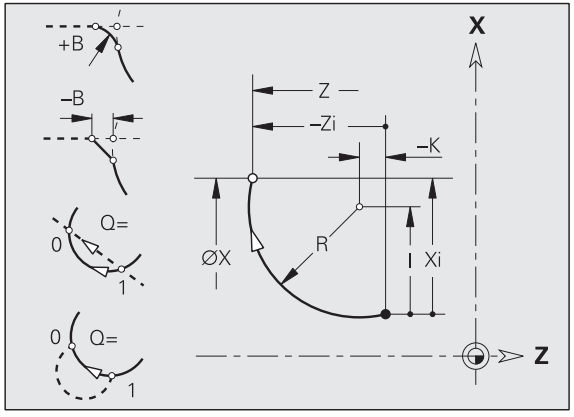
I: Center point incremental (distance from starting point to center as radius)

K: Center point incremental (distance from starting point to center)

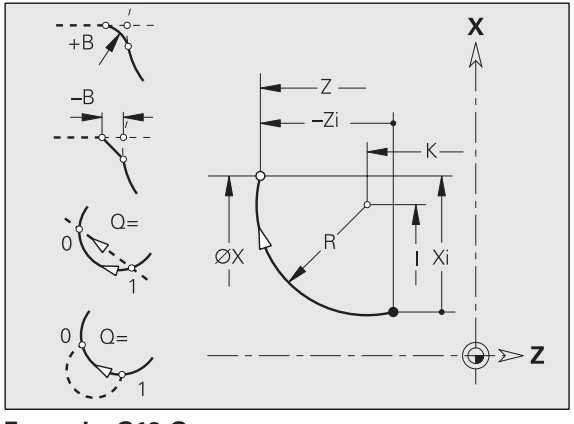
With G12/G13:

I: Absolute center (radius)

K: Absolute center



Example: G2-Geo



Example: G12-Geo

Recess (standard) G22-Geo

G22 defines a recess on an axis-parallel reference element (G1). G22 is assigned to the previously programmed reference element.

Parameters

X: Starting point of recess on the end surface (diameter)

Z: Starting point of recess on lateral surface

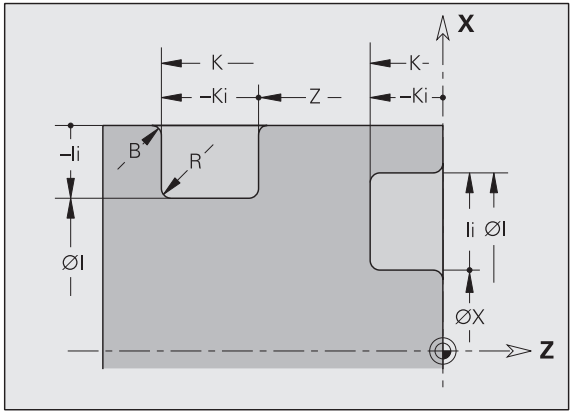
I, K: Inside corner
- I for recess on front face: recess end point (diameter value)
- K for recess on end face: recess base
- I for recess on lateral surface: recess base (diameter value)
- K for recess on lateral surface: recess end point

Ii, Ki: Inside corner – incremental (pay attention to sign !)
- Ii for recess on end face: recess width
- Ki for recess on end face: recess depth
- Ii for recess on lateral surface: recess depth
- Ki for recess on lateral surface: end point of recess (recess width)

B: Outside radius/chamfer (at both ends of the recess) – default: 0
- B>0: Radius of the rounding
- B<0: Width of the chamfer

R: Inside radius (in both corners of recess) – default: 0

Program either X or Z.



Recess (general) G23-Geo

G23 defines a recess on a linear reference element (G1). G23 is assigned to the previously programmed reference element. On the lateral surface the recess can be positioned on an inclined reference straight.

Parameters

H: Recess type – default: 0
- H=0: symmetrical recess
- H=1: free rotation

X: Center point of recess on end surface (diameter)

Z: Center point of recess on lateral surface

I: Recess depth and position
- I>0: recess to right of reference element
- I<0: recess to left of reference element

K: Recess width (without chamfer/rounding)

U: Recess diameter (diameter of recess floor) – use only if the reference element runs parallel to the Z axis.

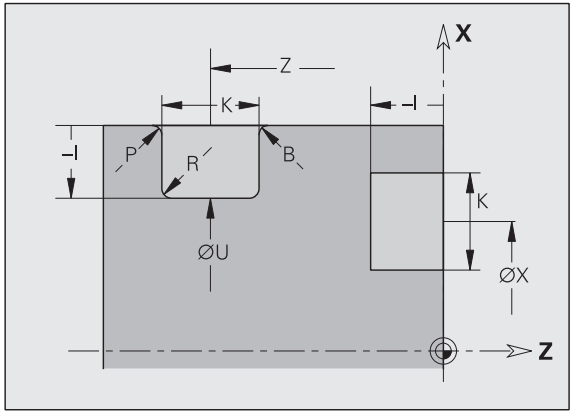
A: Recess angle – default: 0
- with H=0: 0° ≤ A < 180° (angle between edges of recess)
- with H=1: 0° < A ≤ 90° (angle between reference straight and recess edge)

B: Outside radius/corner. Starting point near corner - default: 0
- B>0: Radius of rounding
- B<0: Width of chamfer

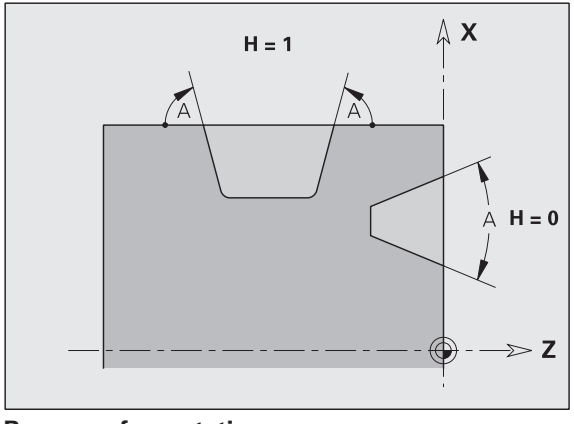
P: Outside radius/corner. Starting point distant from corner - default: 0
- P>0: Radius of rounding
- P<0: Width of chamfer

R: Inside radius (in both corners of recess) – default: 0

The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.



Simple recess



Recess or free rotation

Thread with undercut G24-Geo

G24 defines a linear base element, a linear thread (external or internal thread; metric ISO fine-pitch thread DIN 13 Part 2, Series 1) and a subsequent thread undercut (DIN 76).

Calling the contour macro:

```
N..G1  X..Z..B..     /Starting point for thread
N..G24 F..I..K..Z.. /Contours for thread and undercut
N..G1  X..           /Next surface element
```

Parameters

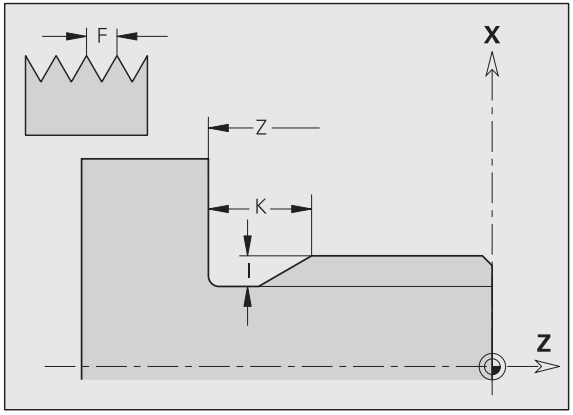
F: Thread pitch

I: Depth of undercut (radius)

K: Width of undercut

Z: End point of the undercut

- G24 can be used only if the thread is cut in the direction of contour definition.
- The thread is machined with G31.



Undercut contour G25-Geo

G25 generates the following undercut contours in paraxial contour corners. The meaning of the parameters depends on the type of undercut.

If you program G25

- **after** the reference element, the undercut is turned at the end of the reference element.
- **before** the reference element, the undercut is turned at the beginning of the reference element.

Calling the contour macro (example):

```
N..G1 Z..              /Linear element as reference
N..G25 H..I..K.. ..    /Undercut contour
N..G1  X..             /Next surface element
```

Parameters

H: Type of undercut – default: 0
- H=4: undercut form U
- H=0, 5: undercut form DIN 509 E
- H=6: undercut form DIN 509 F
- H=7: thread undercut DIN 76
- H=8: undercut form H
- H=9: undercut form K

Undercut form U (H=4)

Parameters

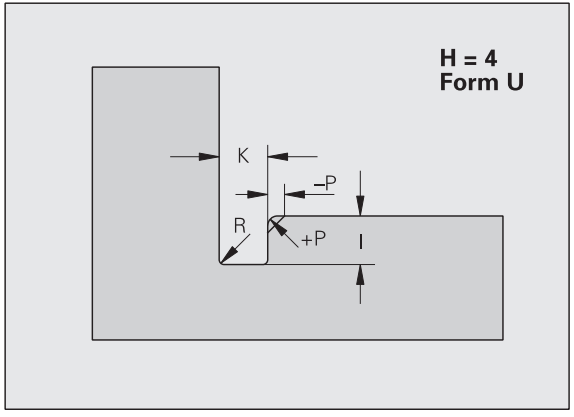
I: Depth of undercut (radius)

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
- P<0: width of the chamfer

Continued ▶



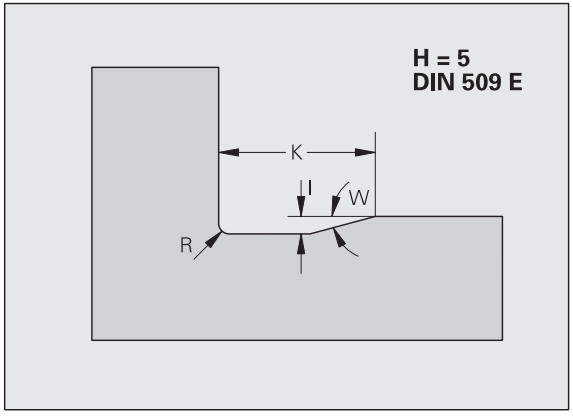
Undercut form U (H=4)

Undercut DIN 509 E (H=0, 5)

Parameters

I: Depth of undercut (radius)
K: Width of undercut
R: Undercut radius (in both corners of the undercut)
W: Undercut angle

If you do not enter any parameters the CNC PILOT calculates the values from the diameter (see User's Manual, section "Undercut Parameters DIN 509 E").



Undercut DIN 509 E (H=0, 5)

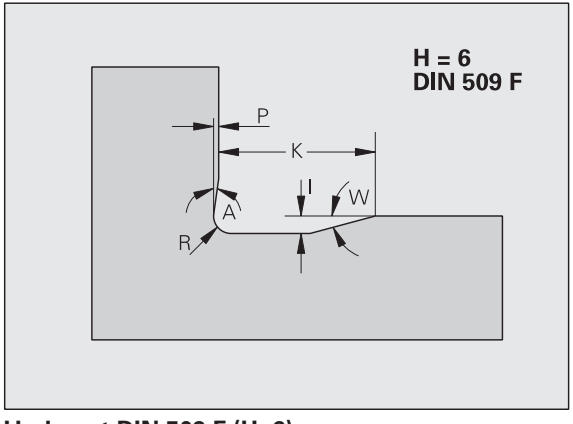
Undercut DIN 509 F (H=6)

Parameters

I: Depth of undercut (radius)
K: Width of undercut
R: Undercut radius (in both corners of the undercut)
P: Transverse depth
W: Undercut angle
A: Transverse angle

If you do not enter any parameters the CNC PILOT calculates the values from the diameter (see User's Manual, section "Undercut Parameters DIN 509 F").

Continued ▶



Undercut DIN 509 F (H=6)

Undercut DIN 76 (H=7)

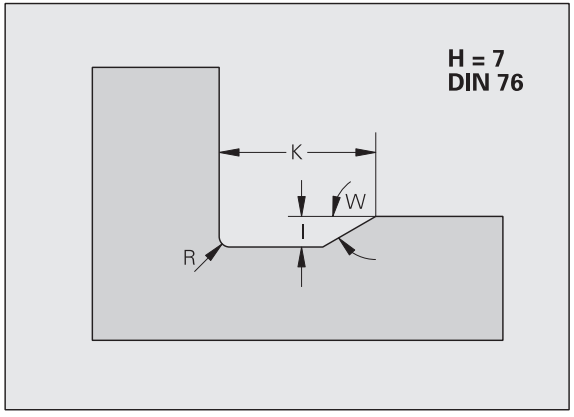
Parameters

I: Depth of undercut (radius)

K: Width of undercut

R: Undercut radius (in both corners of the undercut) – default: R=0.6*I

W: Undercut angle – default: 30°



Undercut DIN 76 (H=7)

Undercut form H (H=8)

If you do not enter W, it will be calculated on the basis of K and R. The final point of the undercut is then located at the "final point contour."

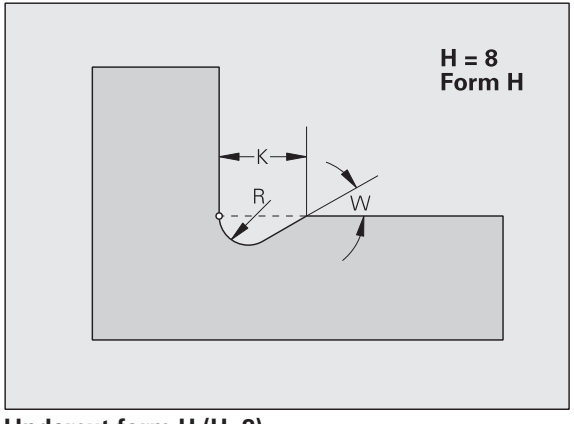
Parameters

K: Width of undercut

R: Undercut radius – no value: the circular element is not machined

W: Plunge angle – no value: W is calculated

Continued ▶



Undercut form H (H=8)

Undercut form K (H=9)

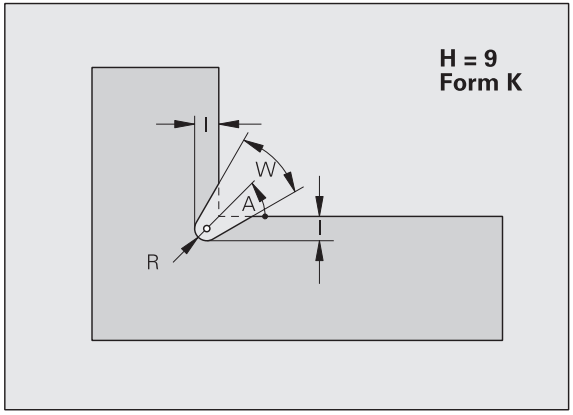
Parameters

I: Undercut depth

R: Undercut radius – no value: the circular element is not machined

W: Undercut angle

A: Angle to linear axis – default: 45°



Undercut form K (H=9)

Thread (standard) G34-Geo

G34 defines a simple or an interlinked external or internal thread (metric ISO fine-pitch thread DIN 13 Series 1). Threads are interlinked by programming several G01/G34 blocks after each other.

Parameters

F: Thread pitch – no value: pitch from the standard table

- You need to program a linear contour element as a reference before G34 or in the NC block containing G34.
- The thread is cut with G31.

Thread (general) G37-Geo

G37 defines the different types of thread. Threads are interlinked by programming several G01/G34 blocks after each other.

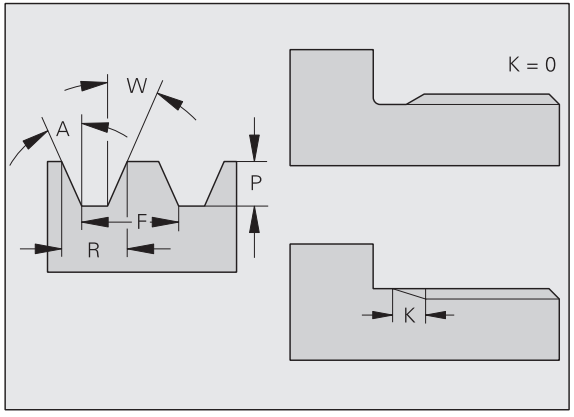
Parameters

Q: Type of thread – default: 1
- Q=1: metric ISO fine-pitch thread (DIN 13 Part 2, Series 1)
- Q=2: metric ISO thread (DIN 13 Part 1, Series 1)
- Q=3: metric ISO taper thread (DIN 158)
- Q=4: metric ISO tapered fine-pitch (DIN 158)
- Q=5: metric ISO trapezoid thread (DIN 103 Part 2, Series 1)
- Q=6: flat metric trapezoid thread (DIN 308 Part 2, Series 1)
- Q=7: metric buttress thread (DIN 13 Part 2, Series 1)
- Q=8: cylindrical round thread (DIN 405 Part 1, Series 1)
- Q=9: cylindrical Whitworth thread (DIN 259)
- Q=10: tapered Whitworth thread (DIN 2999)
- Q=11: Whitworth pipe thread (DIN 2999)
- Q=12: nonstandard thread
- Q=13: UNC US coarse thread
- Q=14: UNF US fine-pitch thread
- Q=15: UNEF US extra-fine-pitch thread
- Q=16: NPT US taper pipe thread
- Q=17: NPTF US taper dryseal pipe thread
- Q=18: NPSC US cylindrical pipe thread with lubricant
- Q=19: NPFS US cylindrical pipe thread without lubricant

F: Thread pitch – must be entered for Q=1, 3..7, 12.

P: Thread depth – enter only for Q=12.

K: Runout length (for threads without undercut) – default: 0



- Program a linear contour element as a reference before G37.
- The thread is cut with G31.
- For standard threads, the parameters P, R, A and W are defined by the CNC PILOT.
- Use Q=12 if you wish to use individual parameters.

The thread is generated to the length of the reference element. For the machining of threads without an undercut, it is necessary to program an additional linear element so that the overrun can be executed by the CNC PILOT without danger of collision.

Continued ▶

D: Reference point (position of thread runout) – default: 0
- D=0: runout at end of reference element
- D=1: runout at beginning of reference element

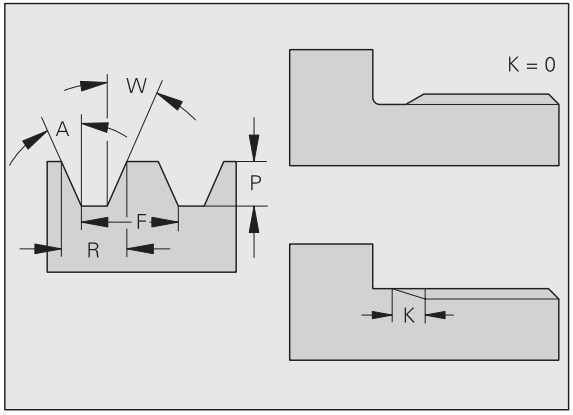
H: Number of grooves – default: 1

A: Edge angle left – enter only for Q=12.

W: Edge angle right – enter only for Q=12.

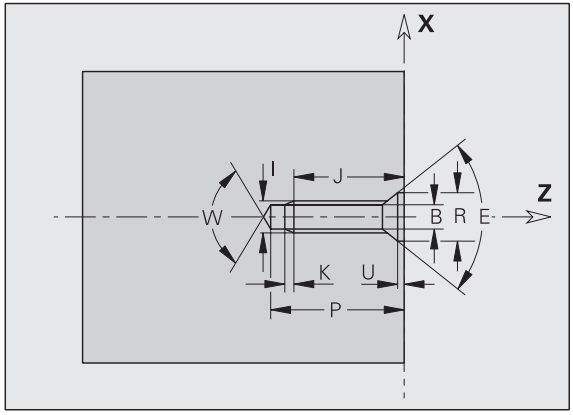
R: Thread width – enter only for Q=12.

E: Variable pitch (increases/reduces the pitch per revolution by E) – default: 0



Bore hole (centered) G49-Geo

G49 defines a single bore hole with countersink and thread **at the turning center** (front or end face).



Parameters

Z: Starting position for hole (reference point)
B: Bore hole diameter
P: Depth of hole (excluding point)
W: Point angle – default: 180°
R: Countersinking diameter
U: Countersinking depth
E: Countersinking 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 (position of bore hole) – default: 0
- A=0: front end
- A=180: tail end

O: Centering diameter

- G49 is programmed in the FINISHED PART section (not in the FRONT or REAR SIDE section).
- The contour defined with G49 is machined with G71...G74.

Overview: Help commands for contour description	
G7	Accurate stop ON
G8	Accurate stop OFF
G9	Accurate stop blockwise
G10	**influences** finishing feed rate for total contour
G38	influences finishing feed rate for basic contour elements block by block
G39	Only for **form elements**: ■ influences finishing feed rate ■ additive compensation values ■ equidistant finishing allowances
G52	Equidistant finishing allowances – blockwise
G95	**defines** finishing feed rate for total contour
G149	additive compensation values for total contour

Accurate stop ON G7-Geo

G7 switches the "precision stop" on modally. In a "precision stop," the CNC PILOT does not start the next block until the "tolerance window" around the end point is reached (for tolerance window, see machine parameters 1106, 1156, ...).

- The NC block containing G7 is also executed with a precision stop.
- "Precision stop" is used for basic contour elements that are executed with G890 or G840.

Precision stop OFF G8-Geo

G8 switches the precision stop off. The block containing G8 is executed **without** a precision stop.

Blockwise accurate stop G9-Geo

G9 activates a precision stop for the NC block in which it is programmed (see also "G7 Geo").

Peak-to-valley height (surface texture) G10-Geo

G10 influences the finishing feed rate of G890 and thus determines the surface roughness of the workpiece.

Basics of programming

- The peak-to-valley height activated with G10 is modal.
- G10 without parameters deactivates peak-to-valley height.
- G95 Geo deactivates peak-to-valley height.
- G10 RH... (without "H") overwrites the valid peak-to-valley roughness block by block.
- G38 Geo overwrites the valid peak-to-valley roughness block by block.

Parameters

H: Type of surface texture (see also DIN 4768)
- H=1: general roughness (profile depth) Rt1
- H=2: average roughness Ra
- H=3: mean roughness Rz

RH: Peak-to-valley roughness (in µm, inches: µinch)

The peak-to-valley height applies only for basic contour elements.

Feed rate reduction factor G38-Geo

G38 defines a special feed rate for G890.

Parameters

E: Special feed-rate factor (0 < E ≤ 1) – default: 1
(special feed rate = active feed rate * E)

Basics of programming

- G38 is a non-modal function.
- G38 is programmed **before** the contouring element for which it is destined.
- G38 **replaces** another special feed rate or a programmed peak-to-valley height.

The "special feed rate" applies only for basic contour elements.

Attributes for superimposed elements G39-Geo

G39 influences the machining of G890 for the **superimposed elements (form elements)**:

- Chamfers/rounding arcs (for connecting base elements)
- Undercuts
- Recesses

Influence on machining:

- Special feed rate
- Peak-to-valley height
- Additive D compensation
- Equidistant oversizes

Parameters

F: Feed per revolution

V: Type of surface texture (see also DIN 4768)
- V=1: general roughness (profile depth) Rt1
- V=2: average roughness Ra
- V=3: mean roughness Rz

RH: Peak-to-valley height (µm, inch mode: µinch)

D: Number of the additive compensation (901 ≤ D ≤ 916)

P: Finishing allowance (radius)

H: (Translation of P) absolute / additive – default: 0
- H=0: P replaces G57/G58 allowances
- H=1: P is added to G57/G58 allowances

E: Special feed-rate factor (0 < E ≤ 1) – default: 1
(special feed rate = active feed rate * E)

Basics of programming

- G39 is a non-modal function.
- G39 is programmed **before** the contour element for which it is destined.
- G50 before a cycle (MACHINING section) switches G39 oversizes for this cycle off.

Only use peak-to-valley height ("V, RH"), finishing allowance ("F") and special feed rate ("E") alternately!

Blockwise finishing allowance G52-Geo

G52 defines an equidistant finishing allowance which is taken into consideration in G810, G820, G830, G860 and G890.

Basics of programming

- G52 is a non-modal function.
- G52 is programmed in the NC block containing the contour element for which it is destined.
- G50 before a cycle (MACHINING section) switches G52 oversizes for this cycle off.

Parameters

P: Finishing allowance (radius)

H: (Translation of P) absolute / additive – default: 0
- H=0: P replaces G57/G58 allowances
- H=1: P is added to G57/G58 allowances

Feed rate per revolution G95-Geo

G95 influences the finishing feed rate of G890.

Basics of programming

- G95 is a modal function
- G10 switches the G95 finishing feed rate off.

Parameters

F: Feed per revolution

- Use peak-to-valley height and finishing feed rate alternatively.
- The G95 finishing feed rate replaces a finishing feed rate defined in the machining program.

Additive compensation G149-Geo

The CNC PILOT manages 16 tool-independent correction values.

To activate the additive correction function, program G149 followed by a „D number“ (for example, G149 D901). ”G149 D900” resets the additive compensation function.

Basics of programming

- Additive compensation is effective from the block in which G149 is programmed.
- An additive compensation remains active until:
  - the next ”G149 D900”
  - the end of the finished part description

Parameters

D: Additive compensation - Default: D900
Range: 900 to 916

Note the direction of contour description!

Start of pocket/island G308-Geo

G308 defines a new reference level/reference diameter for hierarchically nested front face or lateral surface contours.

Parameters

P: Depth for pocket, height for islands

The **algebraic sign of "Depth P"** defines the position of the milling contour:

- P<0: Pocket
- P>0: Island

Section	P	Surface	Milling floor
FRONT END	P<0	Z	Z+P
FRONT END	P>0	Z+P	Z
REAR END	P<0	Z	Z–P
REAR END	P>0	Z–P	Z
CYLINDER SURFACE	P<0	X	X+(P*2)
CYLINDER SURFACE	P>0	X+(P*2)	X

The milling cycles machine from the "surface" toward the "milling floor."

X: Reference diameter from the section code

Z: Reference plane from the section code

P: "Depth" from G308 or from the cycle parameters

End pocket/island G309-Geo

G309 ends a reference level. Every reference plane defined with G308 **must** be ended with G309!

- **Note with "P":** the addition of a negative number reduces the result, and the subtraction of a negative number increases the result.
- **Island:** The area-milling cycles machine the complete area specified in the contour definition. Islands that are defined within this area are not considered.

Starting point of end face contour G100-Geo

G100 defines the starting point of an end face contour.

Parameters

X, C: Starting point in polar coordinates (diameter, starting angle)

XK,YK: Starting point in Cartesian coordinates

Linear segment in end face contour G101-Geo

G101 defines a line segment in an end face contour.

Parameters

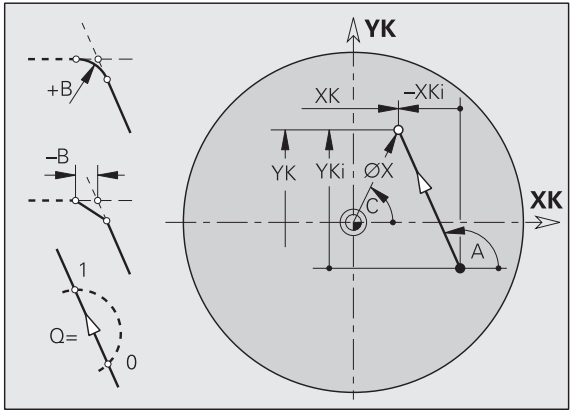
X, C: End point in polar coordinates (diameter, end angle)

XK,YK: End point in Cartesian coordinates

A: Angle to positive XK-axis

B: Chamfer/rounding
- B is undefined: Tangential transition
- B=0: Nontangential transition
- B>0: Rounding radius
- B<0: Chamfer width

Q: Select point of intersection – default: 0
- Q=0: Near intersection
- Q=1: Far intersection



Circular arc in front end contour G102-/G103-Geo

G102/G103 defines a circular arc in a front/end face contour. The direction of rotation is visible in the help graphic.

Parameters

X, C: End point in polar coordinates (diameter, end angle)

XK, YK: End point in Cartesian coordinates

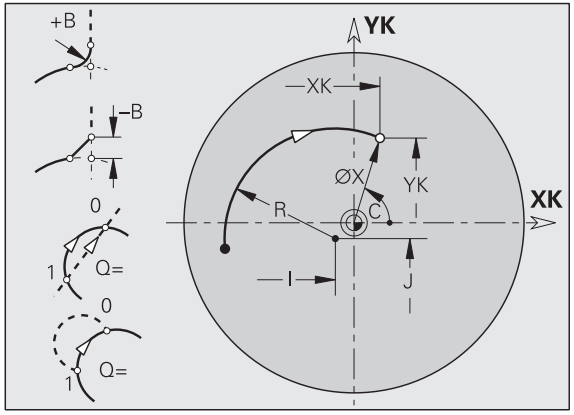
R: Radius

I, J: Center in Cartesian coordinates

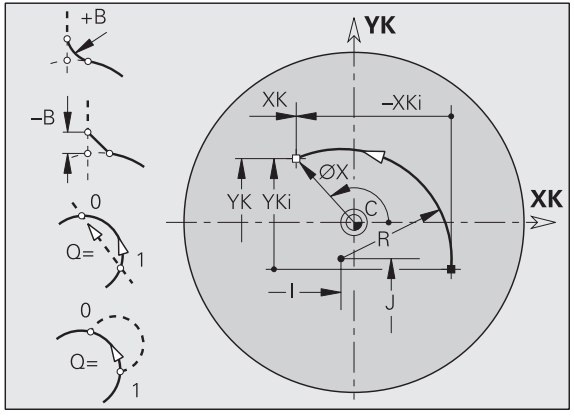
Q: Selection of intersection – default: 0
- Q=0: Far intersection
- Q=1: Near intersection

B: Chamfer/ rounding at end of circular arc
- B no entry: tangential transition
- B=0: no tangential transition
- B>0: Radius of rounding
- B<0: Width of chamfer

The end point may not be the same as the starting point (not a full circle).



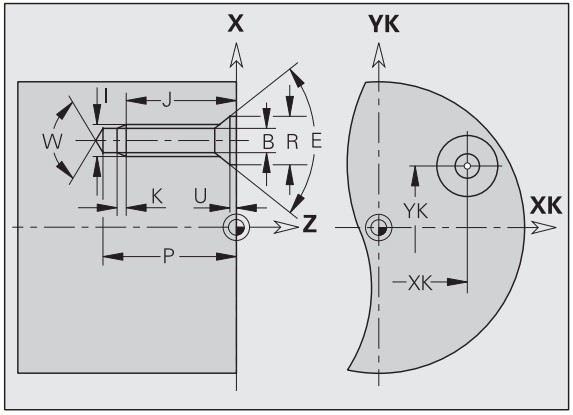
G102-Geo



G103-Geo

Bore hole on end face G300-Geo

G300 defines a bore hole with countersink and thread on the front/end face.



Parameters

XK,YK: Center of hole

B: Hole diameter

P: Depth of hole (excluding point)

W: Point angle – default: 180°

R: Countersinking diameter

U: Countersinking depth

E: Countersinking 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 (reference: Z-axis)
- Front end – default: 0° (range: –90° < A < 90°)
- Rear end – default: 180° (range: 90° < A < 270°)

O: Centering diameter

Use G71...G74 to machine bore holes defined with G300-Geo.

Linear slot on the end face G301-Geo

G301 defines a linear slot on the front/rear end face.

Parameters

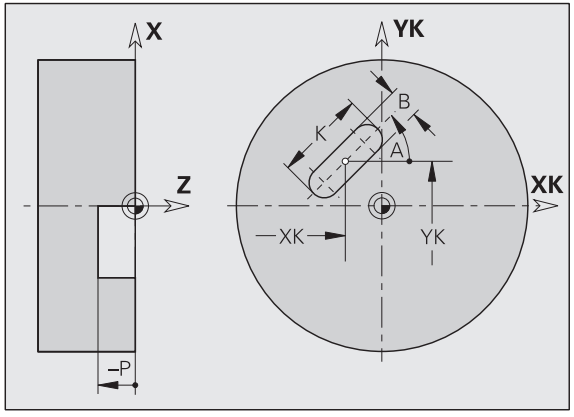
XK,YK: Center of slot

A: Angle of slot length (reference: XK-axis) – default: 0°

K: Slot length

B: Slot width

P: Depth/height – no entry: "P" from G308
- P<0: Pocket
- P>0: Island



Circular slot on the front end G302-/G303-Geo

G302/G303 defines a circular slot on the front/rear end face.

Parameters

I, J: Center of slot curvature

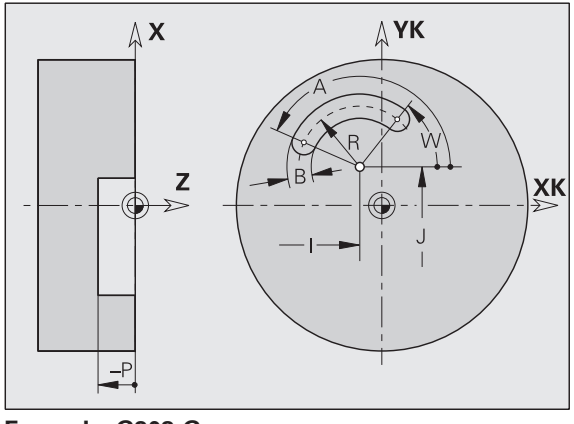
R: Curvature radius (reference: center point path of the slot)

A: Angle of slot starting point (reference: XK-axis) – default: 0

W: Angle of slot end point (reference: XK-axis)

B: Slot width

P: Depth/height – no entry: "P" from G308
- P<0: Pocket
- P>0: Island



Example: G302-Geo

Full circle on front end face G304-Geo

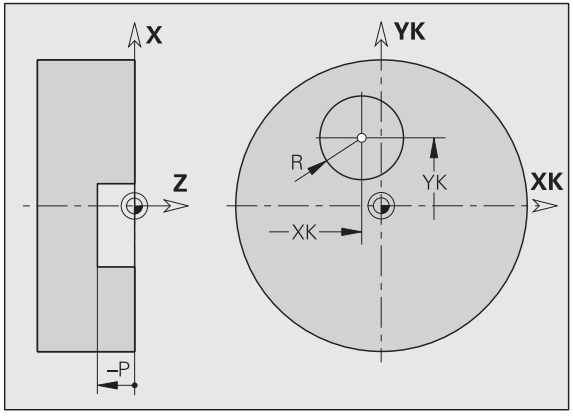
G304 defines a full circle on the front/rear end face.

Parameters

XK,YK: Circle center

R: Circle radius

P: Depth/height – no entry: "P" from G308
- P<0: Pocket
- P>0: Island



Rectangle on end face G305-Geo

G305 defines a rectangle on the front/rear end face.

Parameters

XK,YK: Center of rectangle

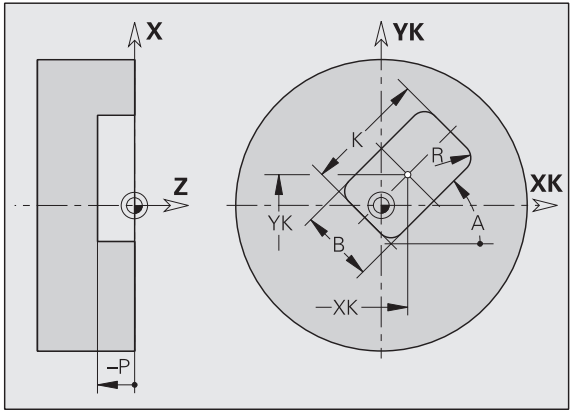
A: Angle to longitudinal axis of rectangle (reference: XK-axis) – default: 0°

K: Length of rectangle

B: (Height) width of rectangle

R: Chamfer/rounding - default: 0
- R>0: Radius of rounding
- R<0: Width of chamfer

P: Depth/height – no entry: "P" from G308
- P<0: Pocket
- P>0: Island



Regular polygon on end face G307-Geo

G307 defines a regular polygon on the front/rear end face.

Parameters

XK,YK: Polygon center point

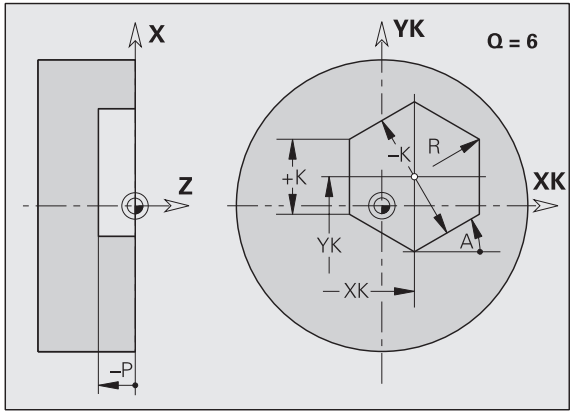
Q: Number of edges (Q≥3)

A: Angle to a polygon side (reference: XK-axis) – default: 0°

K: Edge length
- K>0: Edge length
- K<0: Key width (inside diameter)

R: Chamfer/rounding - default: 0
- R>0: Radius of rounding
- R<0: Width of chamfer

P: Depth/height – no entry: "P" from G308
- P<0: Pocket
- P>0: Island



Linear pattern on end face G401-Geo

G401 defines a linear pattern on the front/rear end face. G401 is effective for the bore hole/figure defined in the following block (G300..305, G307).

Parameters

Q: Number of figures

XK,YK: Starting point of pattern

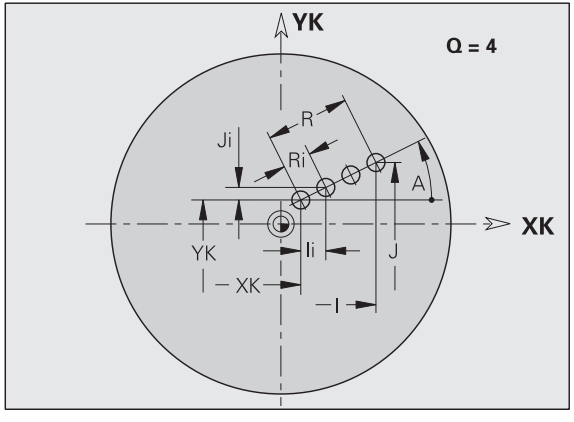
I, J: End point of pattern

Ii, Ji: Distance between two figures (in XK, YK direction)

A: Angle to the longitudinal axis of the pattern (reference: XK-axis)

R: Total length of pattern

Ri: Distance between two figures (pattern distance)



Circular pattern on end face G402-Geo

G402 defines a circular pattern on the front/rear end face. G402 is effective for the figure defined in the following block (G300..305, G307).

Programming notes

- 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.

Parameters

Q: Number of figures

K: Pattern diameter

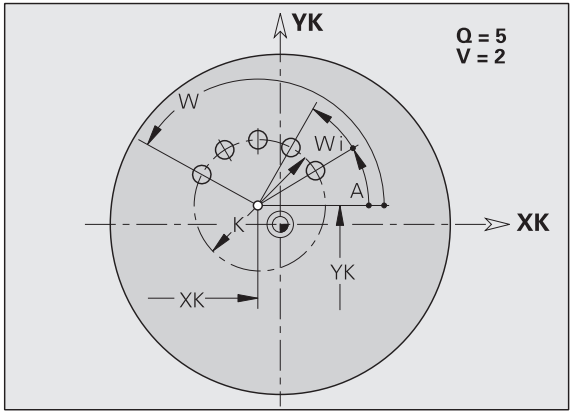
A, W: Starting angle, end angle – position of first/last figure (reference: XK-axis) – default: A=0°; W=360°

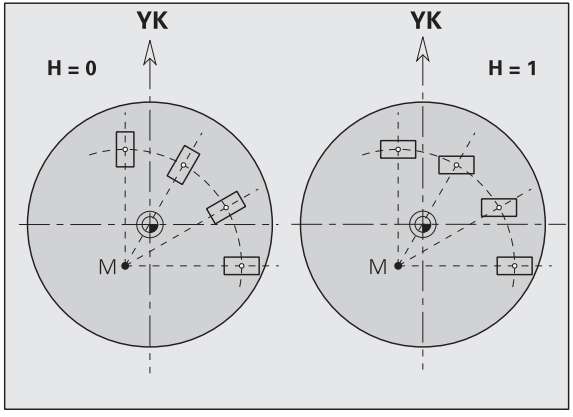
Wi: Angle between two figures

V: Direction (orientation) – default: 0
- V=0 – without W: Distribution over complete circle
- V=0 – with W: Distribution over long arc
- V=0 – with Wi: Algebraic sign of Wi defines the direction (Wi<0: Pattern in clockwise direction)
- V=1 – with W: Pattern in clockwise direction
- V=1 – with Wi: Pattern in clockwise direction (algebraic sign of Wi is without meaning)
- V=2 – with W: Pattern counterclockwise
- V=2 – with Wi: Pattern counterclockwise (algebraic sign of Wi has no effect)

XK,YK: Center of pattern

H: Position of figures – default: 0
- H=0: Normal position – figures are rotated about the circle center (rotation)
- H=1: Original position – position of figure remains the unchanged with respect to the coordinate system (translation)





Starting point of lateral surface contour G110-Geo

G110 defines the starting point of a lateral surface contour.

Parameters

Z, C: Starting point of contour (starting point, starting angle)

CY: Starting angle as linear dimension

Program either Z, C or Z, CY.

Straight line in a lateral surface contour G111-Geo

G111 defines a line segment in a lateral surface contour.

Parameters

Z, C: End point, end angle

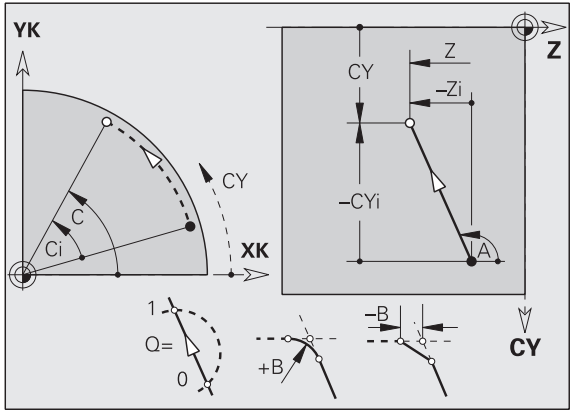
CY: End angle as linear dimension

A: Angle (reference: negative Z axis)

B: Chamfer/rounding
- B is undefined: Tangential transition
- B=0: Nontangential transition
- B>0: Rounding radius
- B<0: Chamfer width

Q: Select point of intersection – default: 0
- Q=0: Near intersection
- Q=1: Far intersection

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. The direction of rotation is visible in the help graphic.

Parameters

Z, C: End point, end angle

CY: End angle as linear dimension

R: Radius

K: Center point (in Z direction)

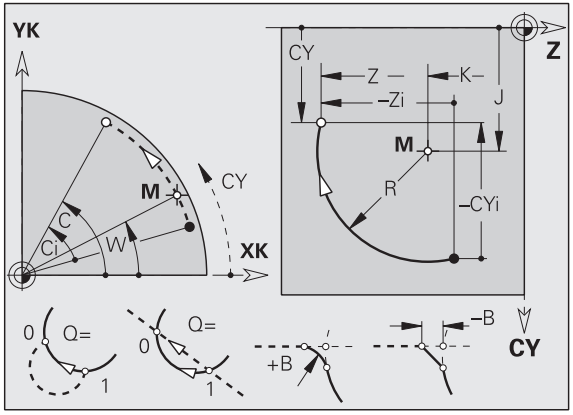
W: Angle of the center point

J: Angle of the center point as a linear value

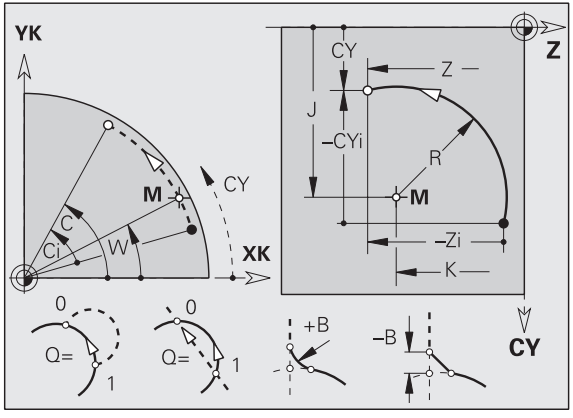
Q: Selection of intersection – default: 0
- Q=0: Far intersection
- Q=1: Near intersection

B: Chamfer/rounding
- B is undefined: Tangential transition
- B=0: Nontangential transition
- B>0: Rounding radius
- B<0: Chamfer width

- Program either Z, C or Z and CY, or K, W or K and J.
- Program either center or radius.
- For radius: only arcs ≤ 180° possible



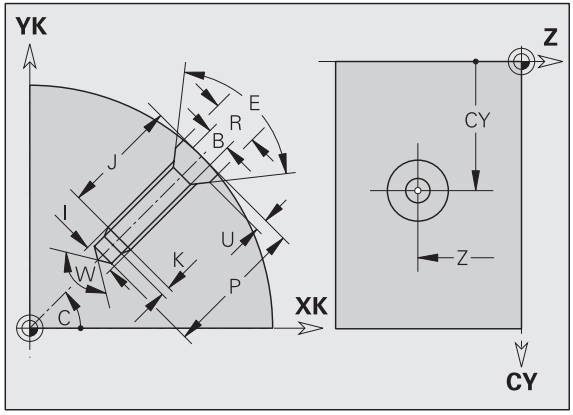
G112-Geo



G113-Geo

Bore hole on lateral surface G310-Geo

G310 defines a bore hole with countersink and thread on the lateral surface.



Parameters

Z, C: Center point of bore hole (position, angle)
CY: Angle as linear dimension
B: Hole diameter
P: Depth of hole (excluding point)
W: Point angle – default: 180°
R: Countersinking diameter
U: Countersinking depth
E: Countersinking 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 (reference: Z axis) – default: 90° = vertical hole (range: 0° < A < 180°)
O: Centering diameter

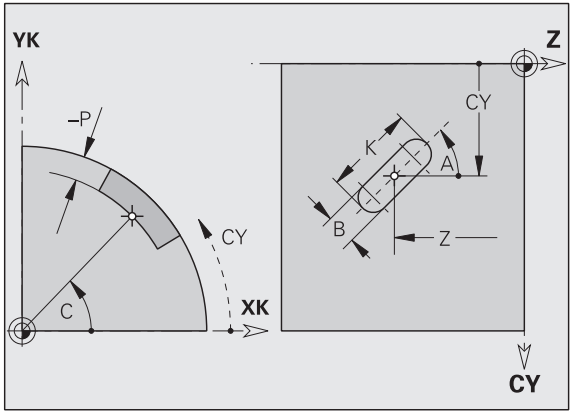
Use G71...G74 to machine contours defined with G310.

Linear slot on lateral surface G311-Geo

G311 defines a linear slot on the lateral surface.

Parameters

Z, C: Center point of slot (position, angle)
CY: Angle as linear dimension
A: Angle to longitudinal axis of slot (reference: Z-axis) – default: 0°
K: Slot length
B: Slot width
P: Pocket depth – no entry: "P" from G308

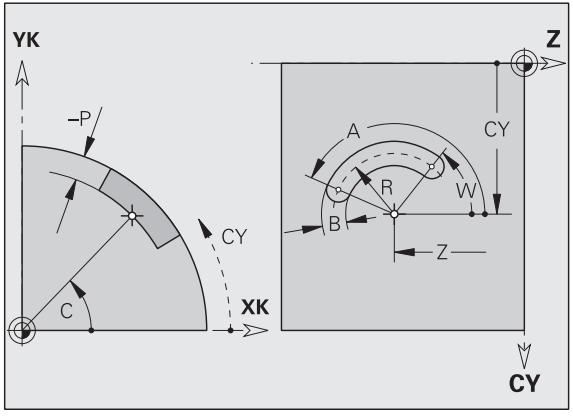


Circular groove on lateral surface G312-/G313-Geo

G312/G313 defines a circular slot on the lateral surface.

Parameters

Z, C: Center of curvature of slot (position, angle)
CY: Angle as linear dimension
R: Curvature radius (reference: center point path of the slot)
A: Angle of slot starting point (reference: Z axis)
W: Angle of slot end point (reference: Z axis)
B: Slot width
P: Pocket depth – no entry: "P" from G308



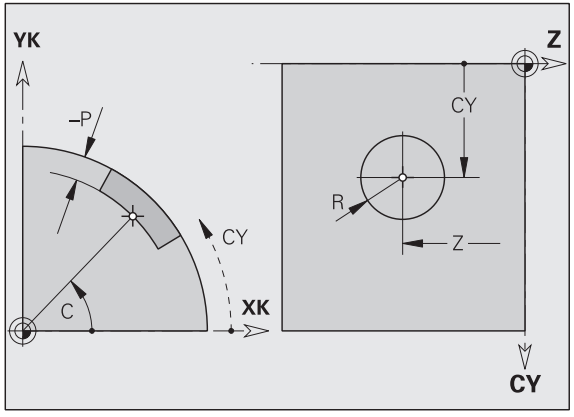
Example: G312-Geo

Full circle on lateral surface G314-Geo

G314 defines a full circle on the lateral surface.

Parameters

Z, C: Circle center (position, angle)
CY: Angle as linear dimension
R: Circle radius
P: Pocket depth – no entry: "P" from G308



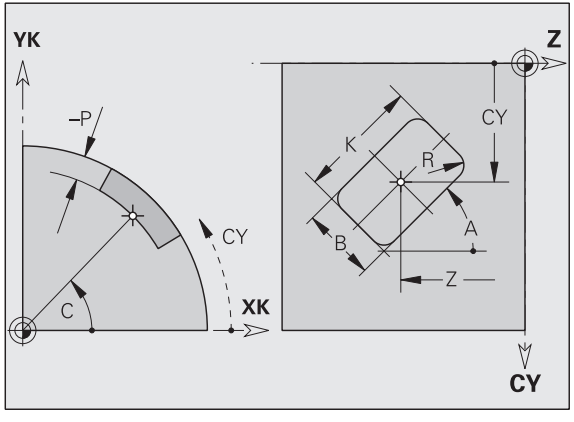
Rectangle on lateral surface G315-Geo

G315 defines a rectangle on the lateral surface.

Parameters

Z, C: Center point of rectangle (position, angle)
CY: Angle as linear dimension
A: Angle to longitudinal axis of rectangle (reference: Z-axis) – default: 0°
K: Length of rectangle
B: Width (height) of the rectangle
R: Chamfer/rounding - default: 0
- R>0: Radius of rounding
- R<0: Width of chamfer

P: Pocket depth – no entry: "P" from G308



Regular polygon on lateral surface G317-Geo

G317 defines a regular polygon on the lateral surface.

Parameters

Z, C: Center point of polygon (position, angle)

CY: Angle as linear dimension

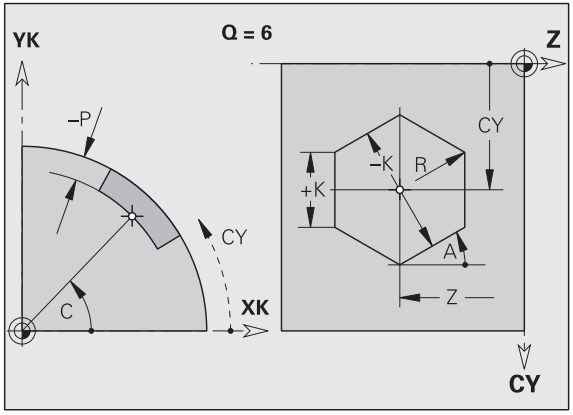
Q: Number of edges (Q≥3)

A: Angle to a polygon side (reference: Z-axis) – default: 0°

K: Edge length
- K>0: Edge length
- K<0: Key width (inside diameter)

R: Chamfer/rounding - default: 0
- R>0: Radius of rounding
- R<0: Width of chamfer

P: Pocket depth – no entry: "P" from G308

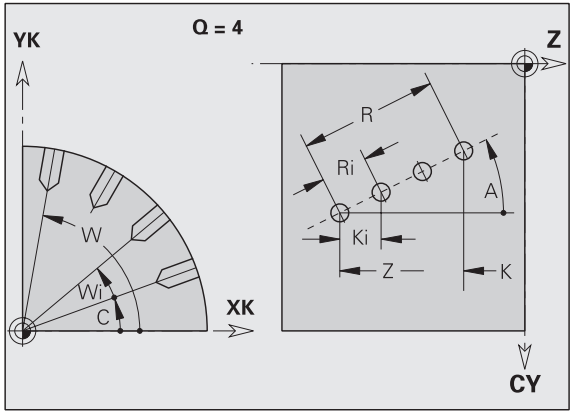


Linear pattern on lateral surface G411 Geo

G411 defines a linear pattern on the lateral surface. G411 is effective for the bore hole/figure defined in the following block (G310..315, 317).

Parameters

Q: Number of figures
Z, C: Starting point, starting angle of pattern
K, W: End point, end angle of pattern
Ki: Distance between the figures (in Z-direction)
Wi: Angular distance between the figures
A: Angle of pattern length axis (reference: Z axis)
R: Total length of pattern
Ri: Distance between two figures (pattern distance)



If you program Q, Z and C, the bore holes/figures will be ordered in a regular manner along the circumference.

Circular pattern on lateral surface G412 Geo

G412 defines a circular pattern on the lateral surface. G412 is effective for the bore hole/figure defined in the following block (G310..315, 317).

Programming notes

- 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.

Parameters

Q: Number of figures

K: Diameter of circle

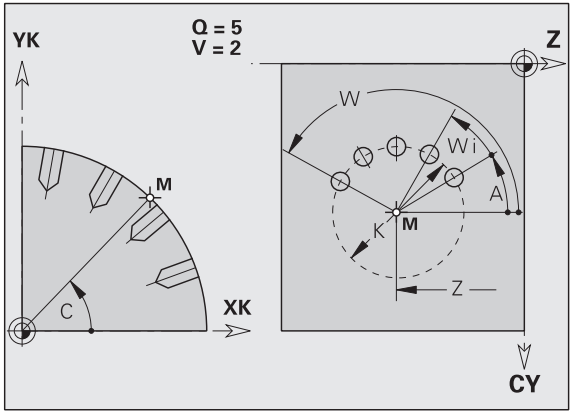
A, W: Starting angle, end angle – position of first/last figure (reference: Z-axis) – default: A=0°; W=360°

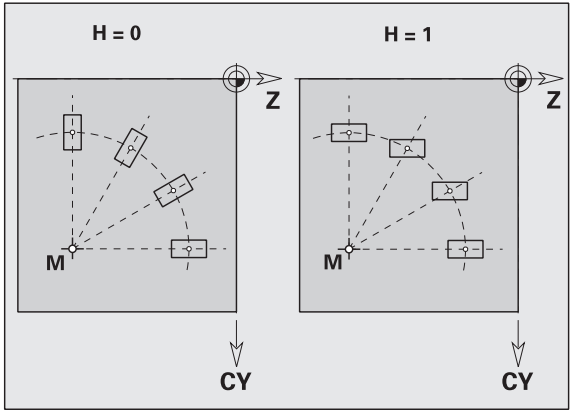
Wi: Distance between two figures

V: Direction (orientation) – default: 0
- V=0 – without W: Distribution over complete circle
- V=0 – with W: Distribution over long arc
- V=0 – with Wi: Algebraic sign of Wi defines the direction (Wi<0: Pattern in clockwise direction)
- V=1 – with W: Pattern in clockwise direction
- V=1 – with Wi: Pattern in clockwise direction (algebraic sign of Wi is without meaning)
- V=2 – with W: Pattern counterclockwise
- V=2 – with Wi: Pattern counterclockwise (algebraic sign of Wi has no effect)

Z, C: Position, angle to midpoint of pattern

H: Position of figures – default: 0
- H=0: Normal position – figure is rotated about the circle center (rotation)
- H=1: Original position – position of figure remains the unchanged with respect to the coordinate system (translation)





G Functions for Machining

The following functions must be preceded with the section code MACHINING.

For **contour-based turning cycles** define the reference to the contour description in "NS, NE." You can determine this reference as follows:

- Activate contour display (menu item "Graphic").
- Place the cursor on the "NS/NE" field and press the Continue key.
- Select the desired contour element with the left/right arrow key.
- The up/down arrow keys can be used to switch between several contours (also end face contours, etc.)
- Transfer the block number of the contour element ("Enter" key).

If you activate the up/down arrow keys, the CNC PILOT also considers contours that are not displayed on the screen.

Tool positioning without machining		Page
G0	Positioning in rapid traverse	45
G14	Approach to the tool change position	45
G701	Rapid traverse to machine coordinates	46

Simple Linear and Circular Paths		Page
G1	Linear path	46
G2	Circular path	47
G3	Circular path	47
G12	Circular path	47
G13	Circular path	47

Feed rate and spindle speed		Page
Gx26	Speed limit *	48
G48	Acceleration (slope)	48
G64	Discontinuous feed rate	49
Gx93	Feed rate per tooth *	49
G94	Feed per minute	49
Gx95	Feed in revolutions	49
Gx96	Constant surface speed	49
Gx97	Spindle speed	49

Cutter radius compensation (TRC/MCRC)		Page
G40	Switch off TRC/MCRC	50
G41	TRC/MCRC left	50
G42	TRC/MCRC right	50

* "x" = number of the spindle (0...3)

Cylindrical surface machining		Page
G110	Rapid traverse, cylindrical surface	85
G111	Linear movement, cylindrical surface	85
G112	Circular arc, cylindrical surface	86
G113	Circular arc, cylindrical surface	86
G120	Reference diameter for cylinder surface machining	82

Milling Cycles		Page
G840	Contour milling	87
G845	Pocket milling – roughing	89
G846	Pocket milling – finishing	89

Other G Functions		Page
G4	Dwell time	90
G7	Accurate stop ON	90
G8	Accurate stop OFF	90
G9	Block precision stop	90
G60	Deactivate protection zone	90
G65	Chucking equipment for graphics	91
G98	Assignment spindle – workpiece	91
G121	Contour mirroring/shifting	54
G702	Regeneration of contour storing/loading	92
G703	Regeneration of contour ON/OFF	92
G720	Spindle synchronization	92
G905	Measuring C-angle offset	92
G918	Feedforward control ON/OFF	93
G919	Spindle override 100%	93
G920	Deactivate datum shift	93
G921	Deactivate datum shift and tool dimensions	93

Other G Functions		Page
G980	Activate datum shift	93
G981	Activate datum shift and tool dimensions	93

Subprograms	Page
Subprogram call	94

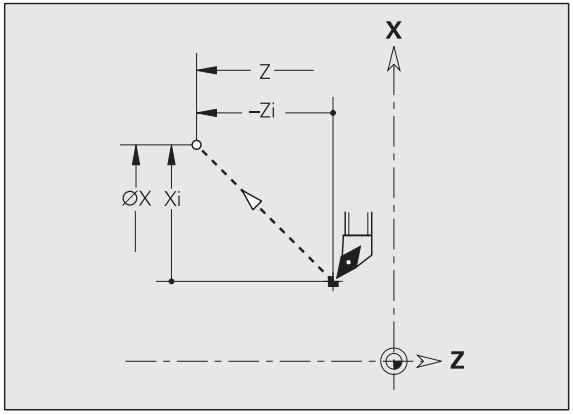
See User's Manual	
G15	Moving a rotary axis
G30	Converting
G62	One-sided synchronization
G63	Synchronous start of slide
G119	Select C axis
G162	Set synchronizing mark
G192	Minute feed rate for rotary axes
G204	Wait for moment
G600	Tool preselection in magazine/chain
G601	Inserting tool from magazine into carrier
G710	Linking tool dimensions
G711	Activate magazine tool
G717	Update nominal values
G718	Servo lag
G900..999	Special G functions

Rapid traverse G0

The tool moves at rapid traverse along the shortest path to the "target point."

Parameters

X, Z: Diameter, length to target point (X diameter)

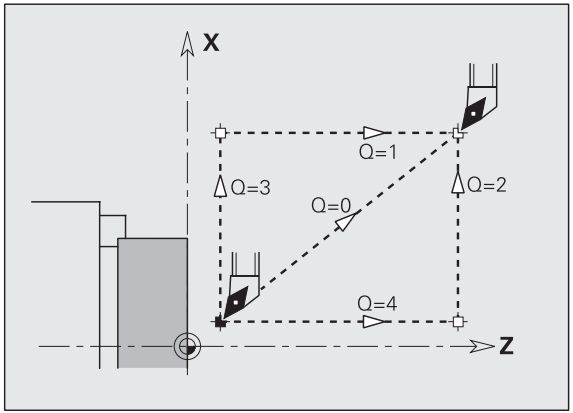


Tool change point G14

The slide moves in rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change.

Parameters

Q: Sequence – default: 0
- 0: Diagonal path of traverse
- 1: First in X axis, then in Z
- 2: First in Z axis, then in X
- 3: X axis only
- 4: Z direction only



Rapid traverse to machine coordinates G701

The slide moves at rapid traverse on the shortest path to the "target point."

Parameters

X, Z: End point (X diameter value)

"X, Z" refer to the machine datum and the slide datum.

Linear path G1

The tool moves at the programmed feed rate on a line to the "end point."

Parameters

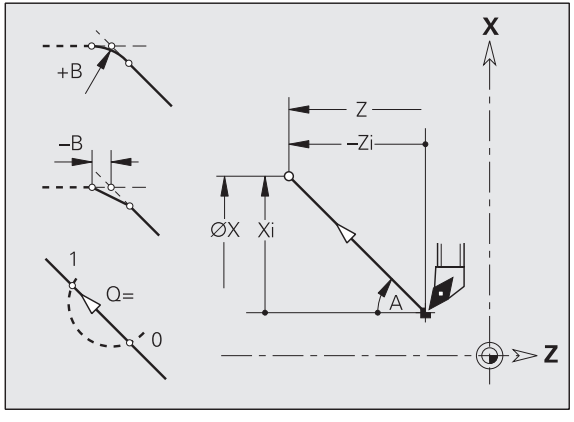
X, Z: Diameter, length to end point (X diameter)

A: Angle (angular direction: see graphic support window)

Q: Selection of intersection – default: Q=0
- Q=0: Near intersection
- Q=1: Far intersection

B: Chamfer/rounding
- B is undefined: Tangential transition
- B=0: Nontangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E: Special feed-rate factor (0 < E ≤ 1) – default: 1
(special feed rate = active feed rate * E)



Circular path

G2, G3 – incremental, G12, G13 – absolute center coordinates

The tool moves in a circular arc at the feed rate to the "end point." The direction of rotation of G2/G3 or G12/G13 is shown in the graphic support window.

The special feed rate applies to chamfer/rounding.

Parameters

X, Z: Diameter, length to end point (X diameter)

R: Radius (0 < R ≤ 200 000 mm)

Q: Selection of intersection – default: Q=0
- Q=0: Far intersection
- Q=1: Near intersection

B: Chamfer/rounding
- B is undefined: Tangential transition
- B=0: Nontangential transition
- B>0: Rounding radius
- B<0: Chamfer width

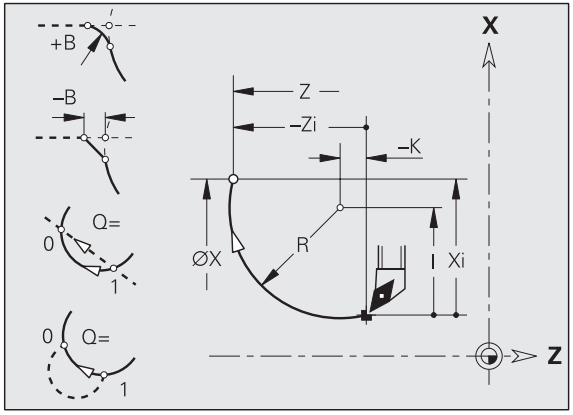
E: Special feed-rate factor (0 < E ≤ 1) – default: 1
(special feed rate = active feed rate * E)

With G2, G3:

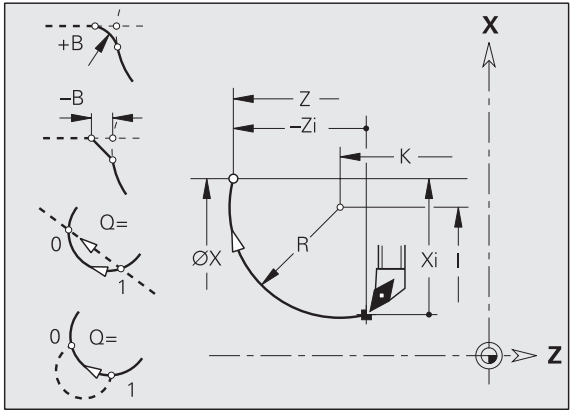
I, K: Center point incremental (distance starting point – center point; I radius)

With G12, G13:

I, K: Absolute center (I radius)



Example: circular path G2



Example: circular path G12

Speed limitation

G26: Spindle; Gx26: Spindle x (x: 1...3)

G26/Gx26 limits the speed of spindle x. The speed limit remains in effect until program end or until it is replaced by a new G26/Gx26.

Parameters

S: (Maximum) speed

The "absolute maximal speed" is specified in machine parameter 805 and following. If S > parameter value, the parameter value takes applies.

Acceleration (slope) G48

G48 specifies the approaching acceleration, braking acceleration, and the maximum feed rate. G48 is a modal function.

Without G48 the following parameter values apply:

- Acceleration at approach/deceleration at departure: Machine parameter 1105 ff "Acceleration/deceleration of linear axis"
- and the maximum feed rate from: Machine Parameter 1101 ff "Maximum axis speed"

Parameters

E: Acceleration at approach – no input: the parameter values take effect.

F: Deceleration at departure – no input: the parameter values take effect

H: Programmed acceleration On/Off
- H=0: switch off programmed acceleration after next traverse
- H=1: switch on programmed acceleration

P: Maximum feed rate – no input: Parameter value

- If P > parameter value, the parameter value applies.
- "E, F and P" refer to the X or Z axis. The acceleration/feed rate for the slide is not higher with axis-parallel traverses.

Interrupted (intermittent) feed G64

G64 interrupts the programmed feed for a short period of time.

- For switch-on, program G64 with E and F.
- For switch-off, program G64 without parameters.

Parameters

E: Duration of pause (range: 0.01 s < E < 99.99 s)

F: Duration of feed rate (range: 0.01 s < E < 99.99 s)

Feed rate per tooth spindle x Gx93

Gx93 (x: spindle 1...3) defines the feed rate **drive-dependent,** depending on the number of teeth of the cutter.

Parameters

F: Feed rate per tooth (mm/tooth or inch/tooth)

The actual value display shows the feed rate in mm/rev.

Feed rate constant G94 (minute feed)

G94 defines the feed rate **as a function of drive.**

Parameters

F: Feed rate per minute (in mm/min or inch/min)

Feed rate per revolution Gx95

G95: Spindle; Gx95: Spindle x (x: 1...3)

G95/Gx95 defines the feed rate **as a function of drive.**

Parameters

F: Feed rate in mm/revolution or inch/revolution

Constant cutting speed Gx96

G96: spindle; Gx96: spindle x (x: 1...3)

G96/Gx96 defines a constant cutting speed. The spindle speed is dependent on the X position of the tool tip or on the tool diameter (with driven tools).

Parameters

S: Cutting velocity (in m/min or ft/min)

Speed Gx97

G97: spindle; Gx97: spindle x (x: 1...3)

G97/Gx97 defines a constant spindle speed.

Parameters

S: Speed (in revolutions per minute)

Tooth and cutter radius compensation (TRC/MCRC) G40, G41, G42

G40: Switch off TRC/MCRC

- The TRC is effective up to the block before G40.
- In the block with G40, or in the block after G40, only one straight line segment permitted (G14 is not permitted).

G41/G42: switch on TRC/MCRC

- In the block with G41/G42 or after the block with G41/G42, one straight line segment (G0/G1) is to be programmed.
- The TRC/MCRC is included after the next contour element.

G41: Switch on TRC/MCRC – displacement of the tool radius in the direction **left of the contour.**

G42: Switch on TRC/MCRC – displacement of the tool radius in the direction **right of the contour.**

Parameters

Q: Machining plane – default: 0

- Q=0: TRC on the turning plane (X-Z plane)
- Q=1: MCRC on the end face (X-C plane)
- Q=2: MCRC on the cylindrical surface (Z-C plane)
- Q=3: MCRC on the end face (X-Y plane)
- Q=4: MCRC on the cylindrical surface (Y-Z plane)

H: Output (only with MCRC) – default: 0

- H=0: Intersecting areas which are programmed in directly successive contour elements are not machined.
- H=1: The complete contour is machined – even if certain areas are intersecting.

O: Feed rate reduction – default: 0

- O=0: Feed rate reduction active
- O=1: No feed rate reduction

- If the tool radii > contour radii, it can result in tool path loops during CRC/MCRC. **Recommendation:** Use the finishing cycle G890 / milling cycle G840.
- Never select MCRC during a perpendicular approach to the plane.
- Note when calling **subprograms** with active TRC/MCRC: Switch the TRC/MCRC off in the main program if it was switched on in the main program. – Switch the TRC/MCRC off in the subprogram in which it was switched on.

Overview	
G51	■ Relative shift ■ Programmed shift ■ Reference: set workpiece datum
G53, G54, G55	■ Relative shift ■ Shift from parameters ■ Reference: set workpiece datum
G56	■ Additive shift ■ Programmed shift ■ Reference: current workpiece datum
G59	■ Absolute shift ■ Programmed shift ■ Reference: machine datum

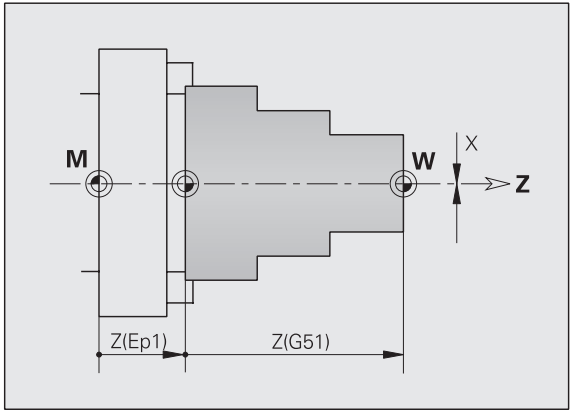
Zero point shift G51

G51 displaces the workpiece datum by "Z" (or "X"). The shift is referenced to the workpiece datum defined in setup operation.

Even if you shift the zero point several times with G51, the displacement is still always referenced to the workpiece zero point from the setup mode.

The zero point shift applies until the end of the program or until it is canceled by another zero point shift.

Parameters
X, Z: Displacement (X radius value) – default: 0



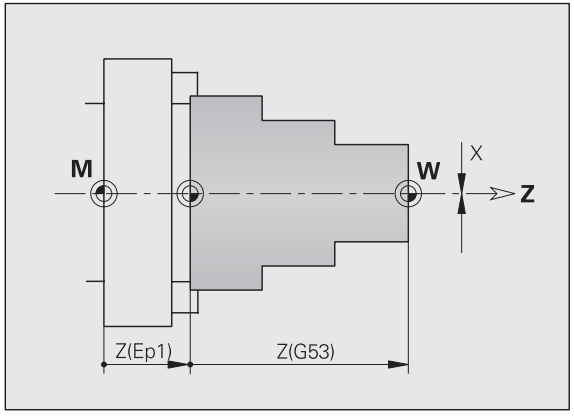
Parameter-dependent zero point shift G53, G54, G55

G53, G54, G55 displace the workpiece zero point by the value defined in the setup parameters 3, 4, 5. The shift is referenced to the workpiece zero point defined in setup operation.

Even if you shift the zero point several times with G53, G54, G55, the displacement is still always referenced to the workpiece datum from the setup mode.

The datum shift applies until the end of the program or until it is canceled by another datum shift.

A shift in X must be indicated as a radius value.

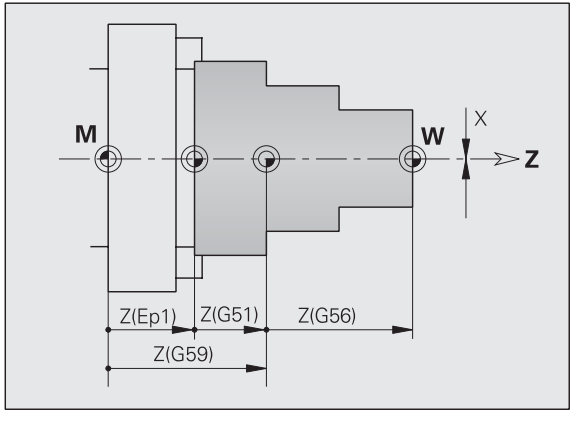


Additive zero point shift G56

G56 shifts the workpiece zero point by "Z" (or "X"). The displacement is referenced to the currently active workpiece zero point.

If you shift the workpiece zero point several times with G56, the displacement is always added to the currently active zero point.

Parameters
X, Z: Shift (X radius value) – default: 0

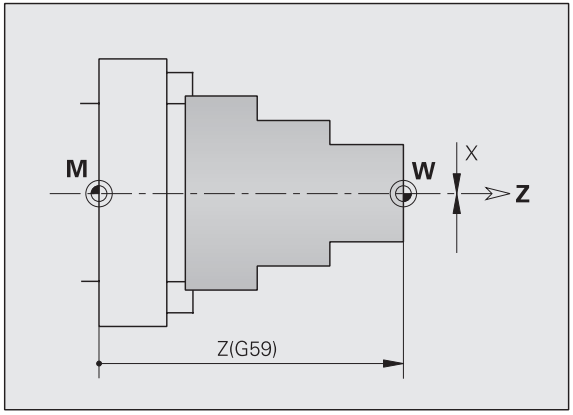


Absolute zero point shift G59

G59 sets the workpiece datum to "X, Z". The new datum is valid to the end of the program.

Parameters
X, Z: Datum shift (X radius dimension)

G59 cancels all previous zero point shifts (with G51, G53..G55 or G59).



G121 contour mirror and shift

G121 mirrors and/or shifts the position of roughed and finished part contours. The contour is mirrored in the X axis and shifted in the Z direction. The workpiece datum is not affected.

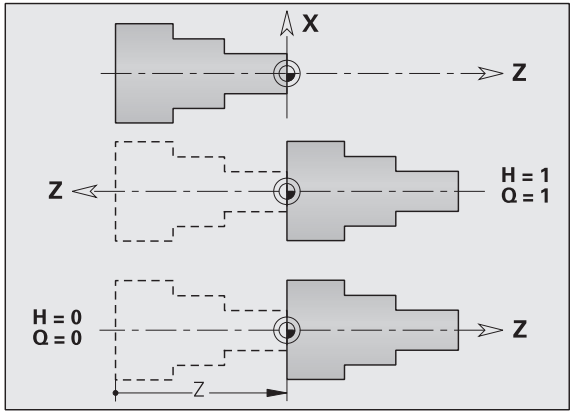
Parameters

H: Mirroring – default: 0
- H=0: Contour shift – no mirroring
- H=1: Contour shift, mirroring, and reversal of the direction of contour description

Q: Mirroring the coordinate system (direction of the Z axis) - default: 0
- Q=0: No mirroring
- Q=1: Mirroring

Z: Shift – default: 0

D: Mirroring XC/XCR (mirroring/shifting the front and rear face contours) - default: 0
- D=0: No mirroring/shifting
- D=1: Mirroring/shifting



- Lateral surface contours are mirrored/shifted like turning contours.
- Auxiliary contours are not mirrored.
- Please note for Q=1: The coordinate system including the contour is mirrored – H=1 mirrors only the contour.

Safety clearance G47

G47 defines the safety clearance for the turning cycles: G810, G820, G830, G835, G860, G869, G890; drilling cycles G71, G72, G74 and milling cycles G840...G846. G47 without parameters activates the parameter values (machining parameters 2, ... – safety clearances).

Parameters

P: Safety clearance

G47 replaces safety clearance set in the machining parameters or that set in G147.

Safety clearance (drilling and milling cycles) G147

G147 defines the safety clearance for milling operations with G840...G846. Parameter "K" influences the safety clearance for drilling operations (G71, G72, G74). G147 replaces the safety clearance set in ";Machining parameter 2, ... – safety clearances."

Parameters

I: Safety clearance in milling direction

K: Safety clearance in approach direction (feed)

G147 replaces safety clearance set in the machining parameters or that set in G47.

Switch off allowance with G50

G50 switches off allowances defined with G52/G39-Geo for the following cycle.

Axis-parallel allowance G57

G57 defines different allowances in X and Z. G57 is programmed before the cycle.

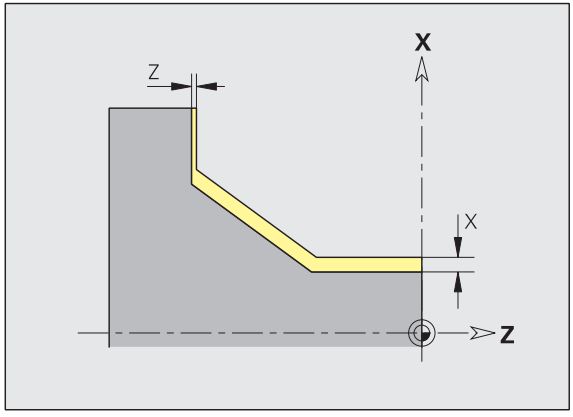
After cycle execution, allowances are

- deleted: G810, G820, G830, G835, G860, G869, G890
- **not** deleted: G81, G82, G83

Parameters

X, Z: Allowance (X diameter value)

If the allowances are programmed with G57 **and** in the cycle itself, the cycle allowances are used.



Contour-parallel allowance (equidistant) G58

G58 defines a contour-parallel allowance. A negative allowance is permitted with G890. G58 is programmed before the cycle.

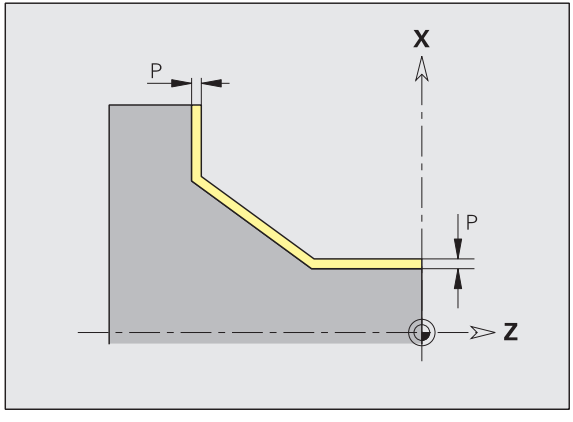
After cycle execution, allowances are

- deleted: G810, G820, G830, G835, G860, G869, G890
- **not** deleted: G81, G82, G83

Parameters

P: Allowance

If an offset is programmed with G58 **and** in the cycle, the offset from the cycle is used.



Tool call T

The "T" is followed by the **WAPP number.** You can enter the T number directly or select it from the tool list (switch with the CONTINUE key).

WAPP number:
W: Tool carrier number
A: Number of the location group
PP: Pocket number in the tool carrier

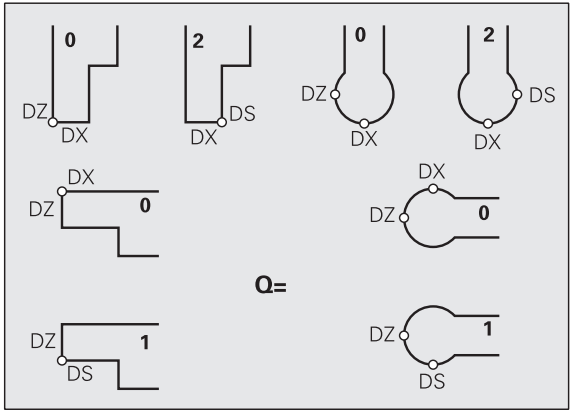
If you are working with **multi tools**, program
"T PP.S" or "T WAPP.S" (S = number of the side tooth).

(Changing the) cutter compensation G148

"O" defines the wear and tear compensation values to be compensated. DX and DZ become effective after program start and after a T command.

Parameters
O: 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 recessing cycles G860, G866 automatically take the "correct" wear compensation into account.



Additive compensation G149

The CNC PILOT manages 16 tool-independent compensation values. To activate the additive compensation function, program G149 followed by a „D number“ (for example, G149 D901). ”G149 D900” resets the additive compensation function.

Parameters

D: Additive compensation - default: D900
Range: 900 to 916

Basics of programming

- The compensation becomes effective only after the tool has actually moved in the compensation direction by the compensation value. You must therefore program G149 one block before the block in which you want the compensation to apply.
- An additive compensation remains effective until:
  - The next ”G149 D900”
  - The next tool change
  - The end of the program

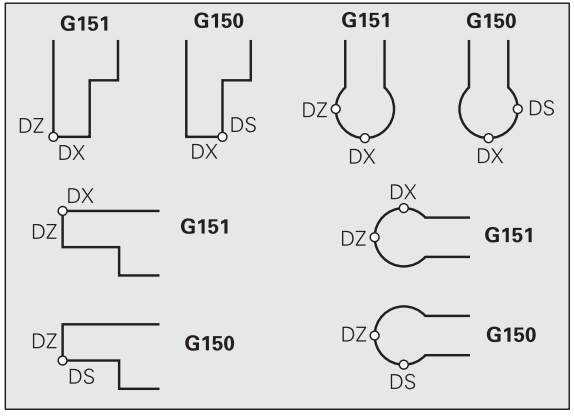
Compensate right tool tip G150
Compensate left tool tip G151

G150/G151 defines the workpiece datum for recessing and button tools.
G150: datum is on right tip
G151: datum 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
- end of the program

- The displayed actual values always refer to the tool tip defined in the tool data.
- Be sure to switch the direction when you work with TRC



End of cycle G80

G80 concludes fixed cycles.

Simple longitudinal roughing G81
Simple face roughing G82

G81/G82 machines (roughs) the contour area described 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.

The tool moves to clearance at the end of the cycle:

- G81: X – last lift-off coordinate; Z – cycle start point
- G82: X – cycle start point; Z – last lift-off coordinate

Parameters

X/Z: Contour target point (X diameter)

Q: G-function Infeed – default: 0
- 0: infeed with G0 (rapid traverse)
- 1: infeed with G1 (feed rate)

G81:

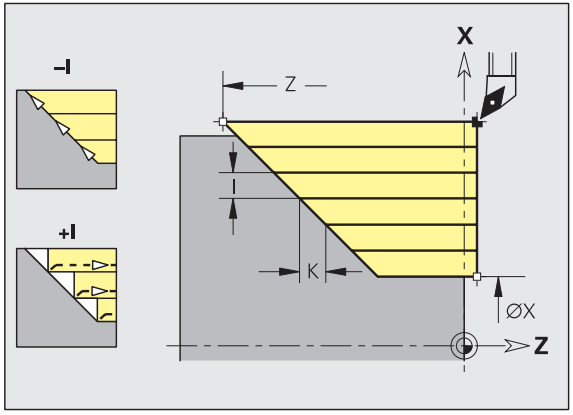
I: Maximum infeed distance in the X direction
- I<0: with machining the contour line
- I>0: without machining the contour line

K: Offset in Z direction – default: 0

G82:

I: Offset in X direction – default: 0

K: Maximum infeed distance in the Z direction
- K<0: with machining the contour line
- K>0: without machining the contour line



Simple longitudinal roughing G81

- **Cutter radius compensation:** is not carried out
- **Offsets (G57):** are calculated and remain effective after end of cycle.
- **Safety clearance** after each step: 1 mm.

Simple contour repeat cycle G83

G83 carries out the functions programmed in the following blocks (simple traverse or cycles without contour description) more than once. G80 ends the machining cycle.

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.

Notes on programming G83

- Stands alone in the block
- Must not be programmed with K variables
- Must not be nested, not even by calling subprograms

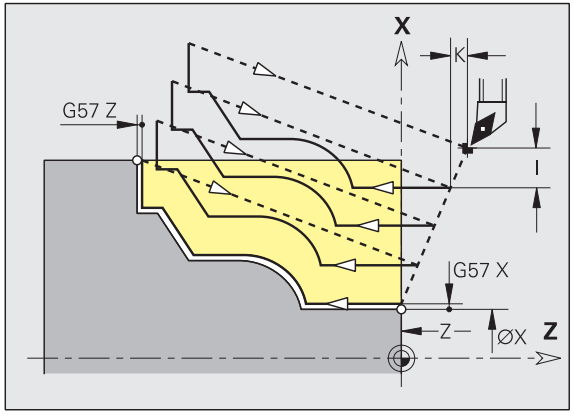
Tool position at end of cycle: Cycle starting point.

Parameters

X/Z: Contour target point (X diameter) - no input: transfer the last X/Z coordinate

I/K: Maximum infeed (I: radius) – default: 0

- **Cutter radius compensation:** is not carried out – You can program the TRC separately with G40..G42.
- **Offsets**: G57-offsets are calculated; G58-offsets only have effect if you are working with TRC. Allowances remain in effect after the end of cycle.
- **Safety clearance** after each step: 1 mm.



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.

Undercut cycle G85

With the function G85, you can machine undercuts according to DIN 509 E, DIN 509 F and DIN 76 (thread undercut). The CNC PILOT decides the type of **undercut** using "K." For undercut parameters, see table.

The adjoining cylinder is machined if the tool is positioned at the cylinder diameter ("X") "in front of" the cylinder.

The undercut roundings are executed with the radius 0.6 * I.

Parameters

X, Z: Destination point (X as diameter value)

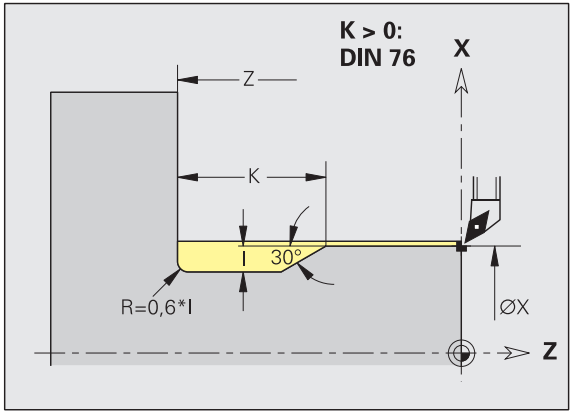
I: Depth/wear allowance (radius)
- DIN 509 E, F: wear allowance – default: 0
- DIN 76: undercut depth

K: Undercut width and **type**
- K no input: DIN 509 E
- K=0: DIN 509 F
- K>0: undercut length for DIN 76

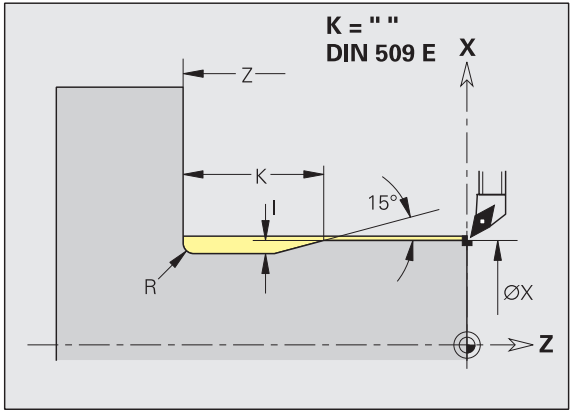
E: Reduced feed (for machining the undercut) – no input: active feed rate

- **Cutter radius compensation:** is not carried out
- **Allowances:** are not considered

Continued ▶



Undercut DIN 76 (thread undercut)



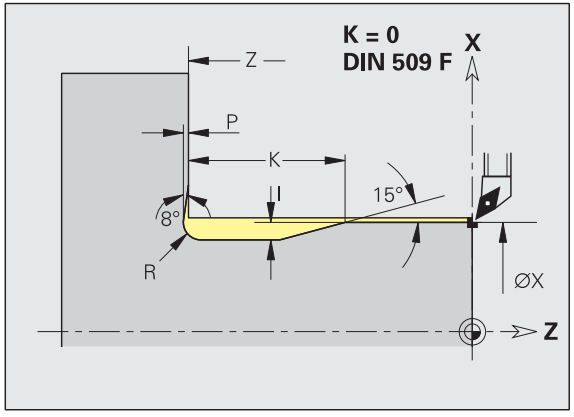
Undercut DIN 509 E

Undercut angle for undercuts according to DIN 509 E and F: 15°
Transverse angle for an undercut according to DIN 509 F: 8°.

where:
I = depth of undercut
K = width of undercut
R = undercut radius
P = depth of end face

Undercut according to 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

Undercut according to DIN 509 F				
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



Undercut DIN 509 F

Recessing G86

G86 machines simple radial and axial recesses with chamfers. From the tool position the CNC PILOT calculates a radial/axial or an inside/outside recess.

If you program an allowance, the control always rough-machines the recess first. In the second step, the recess is then finish-machined.

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. Calculation of the starting position XS (diameter):

XS = XK + 2 * (1,3 – b)

XK: Contour diameter
b: Chamfer width

Tool position at end of cycle:

- Radial recess: X – Starting position; Z – last recess position
- Axial recess: X – last recess position; Z – start position

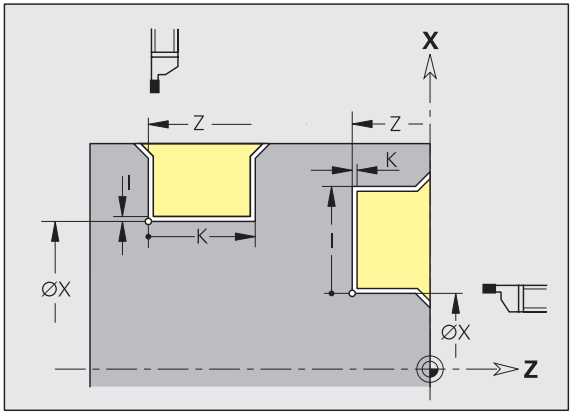
Parameters

X, Z: Target point (X diameter)

Radial recess:

I: Allowance
- I>0: Allowance (roughing and finishing)
- I=0: No finishing

K: Recess width – no input: a single cut is machined (recess width = tool width)



Axial recess:

I: Recess width – no input: a single cut is machined (recess width = tool width)

K: Allowance
- K>0: Allowance (roughing and finishing)
- K=0: No finishing

E Period of dwell (dwell time for chip breaking) – default: Duration of on revolution
- With finishing allowance: Only for finishing
- Without finishing allowance: for every recess

- **Cutter radius compensation:** is not carried out
- **Allowances** are not considered.

Transition radii G87

G87 machines transition radii at orthogonal, paraxial inside and outside corners. The direction for the rounding is taken from the "position/ machining direction" of the tool.

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.

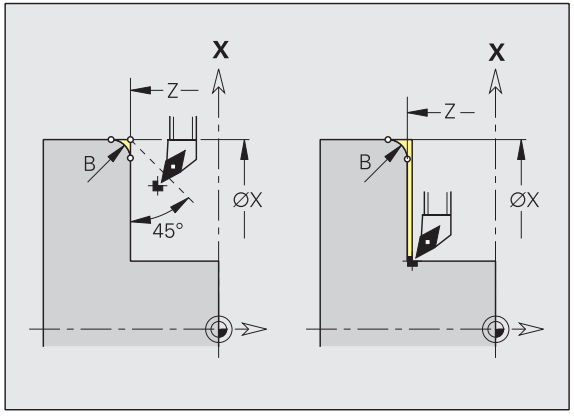
Parameters

X, Z: Corner point (X diameter)

B Radius

E Reduced feed rate – no input: active feed rate

- **Cutter radius compensation:** is carried out
- **Allowances** are not considered.



Chamfer G88

G88 machines chamfers at orthogonal, paraxial outside corners. The direction for the chamfer is taken from the "position/machining direction" of the tool

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.

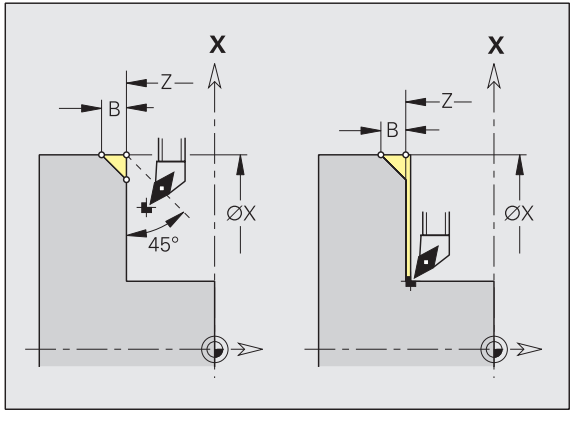
Parameters

X, Z: Corner point (X diameter)

B Chamfer width

E Reduced feed rate – no input: active feed rate

- **Cutter radius compensation:** is carried out
- **Allowances** are not considered.



Longitudinal roughing G810 / Face roughing G820

G810/G820 cuts (roughs) the contour range defined by "NS, NE."

Parameters

NS, NE: Starting block number, end block number (from the contour description)

P: Maximum approach - Maximum infeed distance

I, K: Allowances (I diameter value) – default: 0

E: Approach behavior
- E=0: descending contours are not machined
- E>0: approach behavior
- no input: feed rate reduced depending on approach angle – maximum reduction: 50%

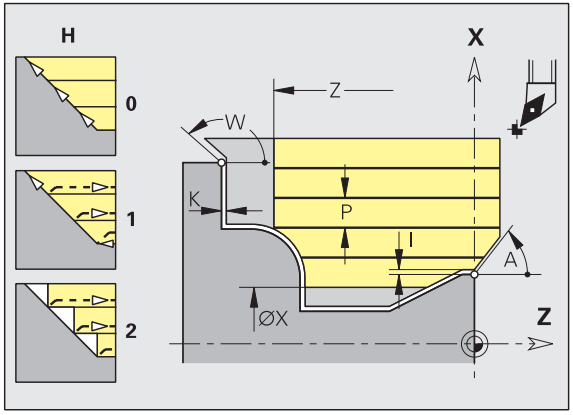
X/Z: Cutting limit (X diameter value)

H: Type of contour smoothing – default: 0
- H=0: smoothing after each cut
- H=1: lift off at under 45°, smoothing after last cut
- H=2: lift off at under 45°, no smoothing

A: Approach angle (reference: Z axis)
- G810 – default: 0°/180° (parallel to Z axis)
- G820 – default: 90°/270° (perpendicular to Z axis)

W: Departing angle (reference: Z axis)
- G810 – default: 90°/270° (perpendicular to Z axis)
- G820 – default: 0°/180° (parallel to Z axis)

Q: Type of retraction after machining – default: 0
n Q=0: return to starting point (G810: first X and then Z direction; G820: first Z and then X direction)
- Q=1: position in front of finished contour
- Q=2: move to clearance height and stop



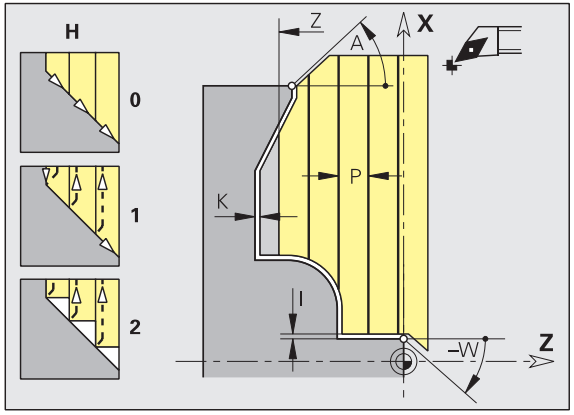
Contour roughing, longitudinal G810

Continued ▶

V: Machining chamfers/roundings at start/end of contour – default: 0
chamfer/rounding is machined:
- V=0: at beginning and end
- V=1: at beginning
- V=2: at end
- V=3: no machining at start or end
- V=4: only single chamfers/roundings are machined – not the base element (requirement: the contour consists of a contour element)

D: Omit element (influences the machining of undercuts, free rotations: see table) – default: 0

B: Slide feed for 4-axis machining
- B=0: Both slides work on the same diameter – at double feed rate
- B<>0: Distance from the "leading" slide (the lead). The slides work at equal feed rate on different diameters.
- B<0: The slides with larger number leads
- B>0: The slide with smaller number leads.



Contour face roughing G820

D	G22	G23	G23	G25	G25	G25
=		H0	H1	H4	H5/6	H7..9
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	–	•	–

„•": Hide elements

- **Cutter radius is compensated**
- **Offsets** (G57/G58): are effective
- On **4-axis cycles,** ensure that the tools are identical (tool type, cutting edge radius, cutting edge angle, etc.).

Contour-parallel roughing G830

G830 machines (roughs) the contour area defined by "NS, NE" parallel to the contour.

Parameters

NS, NE: Starting block number, end block number (from the contour description)

P: Maximum approach - Maximum infeed distance

I, K: Allowances (I diameter value) – default: 0

X/Z: Cutting limit (X diameter value)

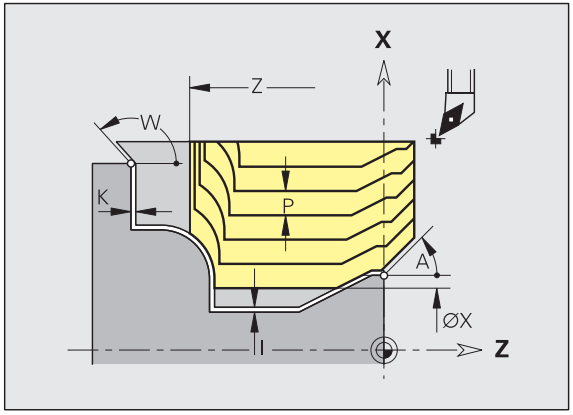
A, W: Approach angle, departure angle (reference: Z axis)
- Approach angle – default: 0°/180° (parallel to Z axis)
- Departure angle – default: 90°/270° (perpendicular to Z axis)

Q: Type of retraction after machining – default: 0
- Q=0: return to starting point (first X and then Z direction)
- Q=1: position in front of finished contour
- Q=2: move to clearance height and stop

V: Machining chamfers/roundings at start/end of contour – default: 0; chamfer/rounding is machined:
- V=0: at beginning and end

D	G22	G23	G23	G25	G25	G25
=		H0	H1	H4	H5/6	H7..9
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	–	•	–

„•": Skipping elements



- V=1: at beginning
- V=2: at end
- V=3: no machining at start or end
- V=4: only single chamfers/roundings are machined – not the base element (requirement: the contour consists of a contour element)

D: Omit element (influences the machining of undercuts, free rotations: see table) – default: 0

- **Cutter radius compensation:** is carried out
- **Offsets** (G57/G58): are effective

Contour-parallel with neutral tool G835

G835 machines the contour area defined by "NS, NE" parallel to the contour and bidirectionally.

Parameters

NS, NE: Starting and end block numbers (reference to contour section)

P: Maximum approach - Maximum infeed distance

I, K: Allowances (I diameter value) – default: 0

X/Z: Cutting limit (X diameter value)

A, W: Approach angle, departure angle (reference: Z axis)
- Approach angle – default: 0°/180° (parallel to Z axis)
- Departure angle – default: 90°/270° (perpendicular to Z axis)

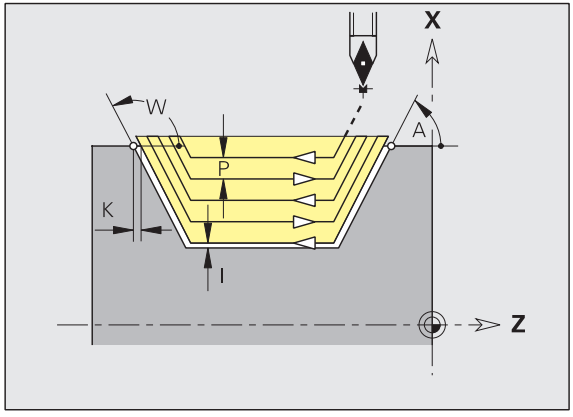
Q: Type of retraction after machining – default: 0
- Q=0: return to starting point (first X and then Z direction)
- Q=1: position in front of finished contour
- Q=2: move to clearance height and stop

V: Machining chamfers/roundings at start/end of contour – default: 0 chamfer/rounding is machined:
- V=0: at beginning and end
- V=1: at beginning
- V=2: at end
- V=3: no machining at start or end
- V=4: only single chamfers/roundings are machined – not the base element (requirement: the contour must consist of one element)

D: Skip the element (influences the machining of undercuts, free rotations: see table) – default: 0

D	G22	G23	G23	G25	G25	G25
=		H0	H1	H4	H5/6	H7..9
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	–	•	–

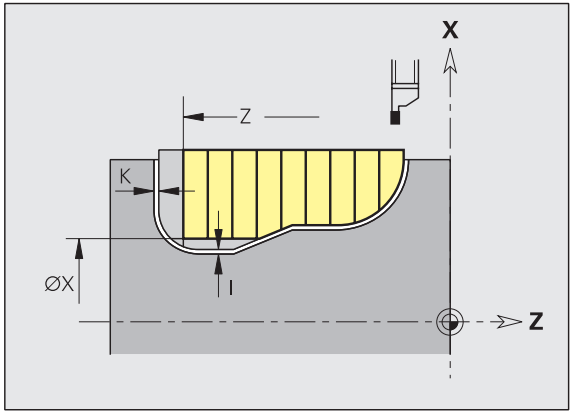
„•": Skipping elements



- **Cutter radius compensation:** is carried out
- **Offsets (G57/G58):** are effective

Recessing G860

G860 machines (indents) the contour area defined by "NS, NE" axially/ radially.



Parameters

NS, NE: Starting block number, end block number (reference on contour section or NS = reference on a recess described with G22-/G23-Geo)

I, K: Allowances (I diameter value) – default: 0

Q: Sequence – default: 0
- Q=0: roughing and finishing
- Q=1: roughing only
- Q=2: finishing only

X/Z: Cutting limit (X diameter value)

V: Machining chamfers/roundings at start/end of contour – default: 0 chamfer/rounding is machined:
- V=0: at beginning and end
- V=1: at beginning
- V=2: at end
- V=3: no machining

E: Finishing feed rate – no input: active feed rate

H: Retraction at end of cycle – default: 0
- H=0: return to starting point (axial recess: first Z and then X direction; radial recess: first X and then Z direction)
- H=1: position in front of the finished contour
- H=2: move to safety clearance and stop

- **Cutter radius compensation:** is carried out
- **Offsets (G57/G58):** are effective

Recessing cycle G866

G866 makes a recess defined with G22-Geo

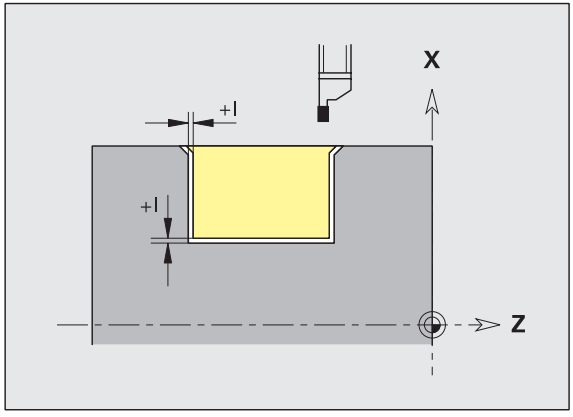
Calculation of the cut segmentation (SBF: see machining parameter 6):
Maximum offset = SBF * cutting edge width

Parameters

NS: Block number (reference to G22-Geo)

I: Allowance (with precutting) – default: 0
- I=0: recess is machined in one run
- I>0: precutting is carried out in the first run; and finishing in the second

E: Dwell time – no input: corresponds to a spindle revolution
- With I=0: for each recess
- With I>0: only during finishing



- **Cutter radius compensation:** is carried out
- **Allowances:** are not considered

Recess turning cycle G869

G869 machines the contour described by "NS, NE" axially and radially with alternating recessing and roughing movements.

Parameters

NS, NE: Starting block number, end block number (from the contour description)

P: Maximum approach - Maximum infeed distance

R: Turning depth compensation for finishing – default: 0

I, K: Allowances (I diameter value) – default: 0

X/Z: Cutting limit (X diameter value)

A, W: Approach angle, departure angle – default: opposite from the recessing direction

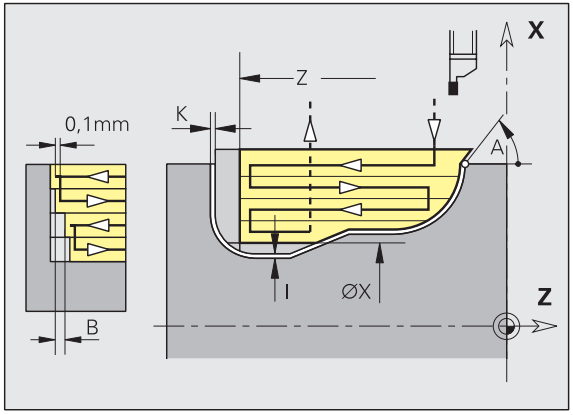
Q: Sequence – default: 0
- Q=0: roughing and finishing
- Q=1: roughing only
- Q=2: finishing only

U: Direction of turning – default: 0
- U=0: Bidirectional turning
- U=1: Unidirectional turning in direction of contour

H: Retraction at end of cycle – default: 0
- H=0: return to starting point (axial recess: first Z and then X direction; radial recess: first X and then Z direction)
- H=1: position in front of the finished contour
- H=2: move to safety clearance and stop

V: Machining chamfers/roundings at start/end of contour – default: 0 chamfer/rounding is machined:
- V=0: At beginning and end
- V=1: At beginning
- V=2: At end
- V=3: No machining

O: Recessing feed rate – default: active feed rate



E: Finishing feed rate – no input: active feed rate

B: Offset width – default: 0

- Cycle G869 requires type 26* tools.
- **Cutter radius compensation:** is performed
- **Offsets** (G57/G58): are effective

Contour finishing G890

G890 finishes the contour area defined by "NS, NE" parallel to the contour.

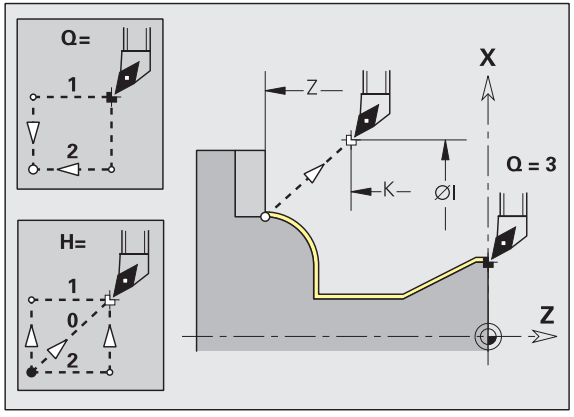
Parameters

NS, NE: Starting block number, end block number (from the contour description)

E: Approach behavior
- E=0: descending contours are not machined
- E>0: approach behavior
- no input: feed rate reduced depending on approach angle – maximum reduction: 50%

V: Machining chamfers/roundings at start/end of contour – default: 0; chamfer/rounding is machined:
- V=0: at beginning and end
- V=1: at beginning
- V=2: at end
- V=3: no machining at start or end
- V=4: only single chamfers/roundings are machined – not the base element (requirement: the contour consists of a contour element)

Q: Type of approach – default: 0
- Q=0: automatic selection – the CNC PILOT checks the possibilities for approach:
  – diagonal approach
  – first X direction, then Z
  – equidistant around the barrier
  – Omission of the first contour element if the start position is inaccessible
- Q=1: First in X direction, then Z
- Q=2: First in Z direction, then X
- Q=3: No approach – tool is in the proximity of the starting point of the contour section
- Q=4: Residual finishing



H: Type of retraction – default: 3
tool lifts off under 45° in the opposite direction to machining and moves as follows to the position "I, K":
- H=0: diagonal
- H=1: first X, then Z direction
- H=2: first Z, then X direction
- H=3: remains at safety clearance
- H=4: no retraction movement – tool remains at the end coordinate

X/Z: Cutting limit (X diameter value) - no input: no limit

D: Omit element (influences the machining of undercuts, free rotations and recesses: see table) – default: 1

Continued ▶

I, K: End point which is approached at the end of the cycle – except for H=3 or 4 (I diameter value)

O: Feed rate reduction – default: 0
- O=0: No feed rate reduction
- O=1: Feed rate reduction active

During **residual finishing** (G890 – Q4), the CNC PILOT, checks whether the tool can access the contour recess without collision. The tool parameter "width dn" is the basis of this collision check.

D	G22	G23	G23	G25	G25	G25	G25
Form		(H0)	(H1)	U	E, F	G, H	K
0	–	–	–	–	–	–	–
1	–	–	•	–	•	•	•
2	–	–	•	–	–	–	–
3	–	–	–	–	•	•	•
4	–	–	•	–	–	•	•
5	–	–	•	–	•	•	–
6	–	–	•	–	•	–	–
7	•	•	•	•	•	•	•

„•": Skipping elements

Other D codes for skipping undercuts/recesses:

G-call	Function	D code
G22	Gasket recess	512
G22	Retaining ring recess	1.024
G23 H0	General recess	256
G23 H1	Free rotation	2.048
G23 H4	Undercut form U	32.768
G23 H5	Undercut form E	65.536
G23 H6	Undercut form F	131.072
G23 H7	Undercut form G	262.144
G23 H8	Undercut form H	524.288
G23 H9	Undercut form K	1.048.576

Thread cycle G31

G31 machines threads defined with G24-Geo, G34-Geo or G37-Geo. G31 also machines successions of threads and multiple threads.

Chamfer length, overrun length: If you do not program "B, P",the CNC PILOT will automatically determine the lengths from adjacent undercuts or axis-parallel recesses. If no undercut/recess exists, the chamfer and overrun lengths from Machining Parameter 7 will be used.

Parameters

NS: Block number (from the finished part description containing the basic block G1-Geo for thread contour definition)

I: Maximum approach - Maximum infeed distance

B, P: Chamfer length, overrun length

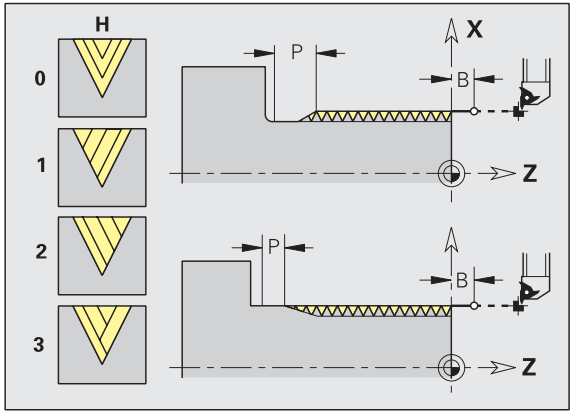
D: Cutting direction (reference: defined direction for basic element) – default: 0;
- D=0: same direction
- D=1: opposite direction

V: Type of feed – default: 0;
- V=0: constant cross section for all cuts
- V=1: constant feed
- V=2: with distribution of remaining cut – if the division thread depth/feed provides a remainder, the first feed is reduced. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
- V=3: without remaining cut division – feed is calculated from pitch and speed

H: Type of offset – default: 0
- H=0: no offset
- H=1: offset to the left
- H=2: offset to the right
- H=3: offset alternating left and right

Q: Number of air cuts after the last cut – default: 0

C: Starting angle – default: 0



- "Feed rate stop" becomes effective only at the end of a thread cut.
- Feed rate override is not effective.
- Do not use the spindle override if the feedforward is switched off!

Danger of collision!
An excessive overrun length P can cause collision. You can check the overrun length in the simulation.

Simple thread cycle G32

G32 cuts a simple thread in any desired direction and position (longitudinal, tapered or transverse thread; internal or external thread). G32 calculates the thread to be cut from the "thread end point," "thread depth" and the current tool position.

Parameters

X, Z: End point of thread (X diameter)

F: Thread pitch

P: Thread depth

I: Cutting depth

B: Remainder cut – default: 0
- B=0: division of the last cut into 1/2, 1/4, 1/8, 1/8 cut.
- B=1: no remaining cut division

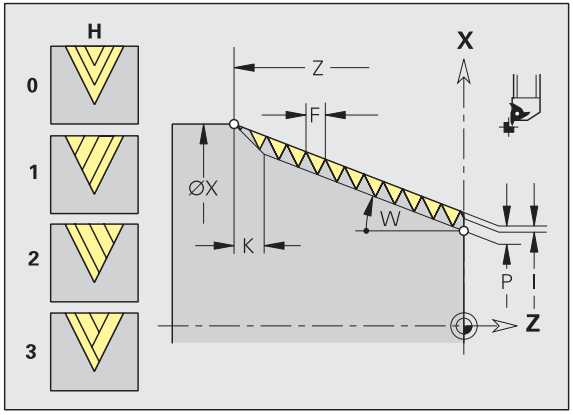
Q: Number of air cuts after the last cut – default: 0

K: Runout length – default: 0 (see G33)

W: Taper angle (range: –45° < W < 45°) – default: 0; position of the taper thread with reference to longitudinal or transverse axis.
- W>0: Rising contour (in machining direction)
- W<0: Falling contour

C: Starting angle – default: 0

H: Type of offset – default: 0
- H=0: no offset
- H=1: offset to the left
- H=2: offset to the right
- H=3: offset alternating left and right



- "Feed rate stop" becomes effective only at the end of a thread cut.
- Feed rate override is not effective.
- Spindle override is not effective.
- Create thread with G95 (feed rate per revolution).
- **Look-ahead control** is switched off.

Individual thread cut G33

G33 cuts threads in any desired direction and position (longitudinal, tapered or transverse threads; internal or external threads). The thread starts at the current tool position and ends at "X, Z".

Parameters

X, Z: Diameter, length to end point of thread (X diameter)

F: Feed rate per revolution (thread pitch)

B, P: Chamfer length, overrun length – default: 0 (see G33)

C: Starting angle – default: 0

Q: Number of the spindle

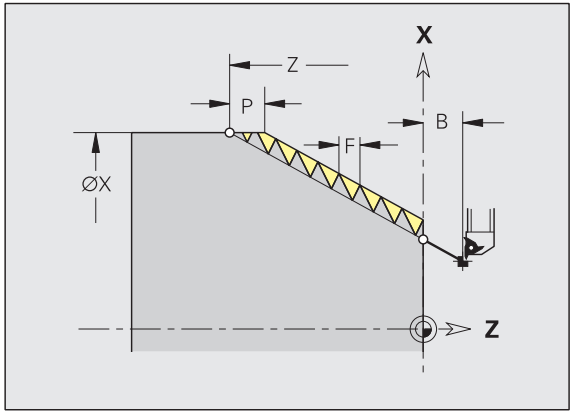
H: Reference direction for spindle pitch - default: 0
- H=0: Feed rate on Z axis (for longitudinal and tapered threads up to a maximum angle of +45°/–45° to Z axis
- H=1: Feed rate in X axis (for transverse and taper threads up to a maximum angle of 45°/–45° to the X axis
- H=2: Feed rate in Y axis
- H=3: contouring feed rate

E: Variable pitch (increases/reduces the pitch per revolution by E) – default: 0

- "Feed rate stop" becomes effective only at the end of a thread cut.
- Feed rate override is not effective.
- Do not use the spindle override if the feedforward is switched off!
- Create thread with G95 (feed rate per revolution).

Calculation of "thread run-in/runout distances"

The slide requires a run-in distance to accelerate to the programmed feed rate before starting the actual thread, and a runout distance at the end of the thread to decelerate again.



Calculation of chamfer (run-in) length:

$BA > 0.75 * (F*S)^2 / a + 0.15$

Calculation of runout length:

$BE > 0.75 * (F*S)^2 / e + 0.15$

BA: Minimum run-in length

BE: Minimum runout length

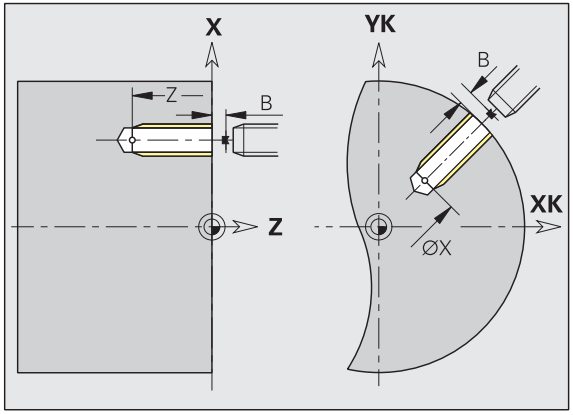
F: Spindle pitch in mm/revolution

S: Speed **in revolutions/second**

a, e: Acceleration in mm/s² (a = "acceleration at start of block"; e = "acceleration at end of block" from Machine Parameter 1105, ff)

Tapping cycle G36

G36 can be used for axial and radial threads using driven or stationary tools. Depending on "X/Z", G36 decides whether a radial or n axial thread will be machined.



Parameters

X: Diameter – end point of thread for axial machining

Z: Length – end point of thread with radial machining

F: Feed per revolution - thread pitch

Q: Number of spindle - default: 0 (main spindle)

B: Chamfer length (see G33)

H: Reference direction for thread pitch – default: 0
- H=0: Feed rate on the Z axis
- H=1: Feed rate on the X axis
- H=2: Feed rate on the Y axis
- H=3: Contour feed rate

S: Retraction speed
default: Same spindle speed as for tapping

- "Cycle STOP" becomes effective only at the end of a thread cut.
- Feed rate override is not effective.
- Do not use spindle override !
- With uncontrolled tool drive, a floating tap holder is required.

Simple drilling cycle G71

G71 is used for axial and radial boring on the end faces or lateral surface using driven or stationary tools.

The cycle is used for:

- A single hole without contour description
- Boring with contour description (single hole or hole pattern)

Parameters

NS: Contour block number with boring geometry (G49, G300- or G310-Geo) – no input: single hole without contour description

X, Z: Position, length – end points of axial/radial hole (X diameter)

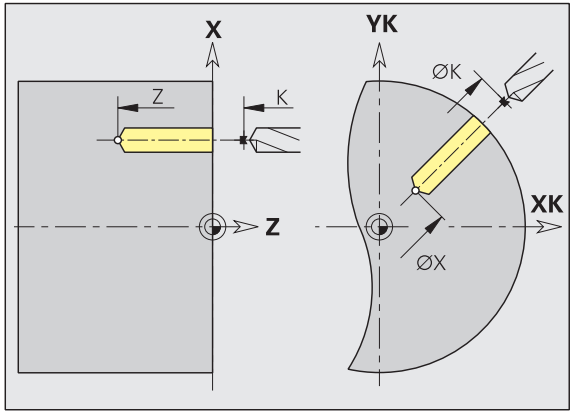
E: Delay time in seconds (dwell time for chip breaking at end of hole) – default: 0

V: Feed rate reduction (50%) – default: 0
- V=0 or 2: Feed rate reduction at start
- V=1 or 3: Feed rate reduction at start and end
- V=4: Feed rate reduction at end
- V=5: No feed rate reduction

Exception for V=0 and V=1: No feed rate reduction when boring/drilling with indexable inserts and twist drills with 180° angle

D: Retraction speed – default: 0
- D=0: Rapid traverse
- D=1: Feed rate

K: Retraction level (radial holes and holes in the YZ plane: Diameter) – No input: Tool moves to starting position or safety clearance



- Single hole without contour description: Program „X or Z“ as alternative.
- Hole with contour description: Do not program „X, Z“.
- Hole pattern: "NS" refers to the bore hole contour (and not the definition of the pattern).

Boring, sinking G72

G72 can be used for boring, sinking, reaming, tapping or centering on the end faces or lateral surface using driven or stationary tools.

Parameters

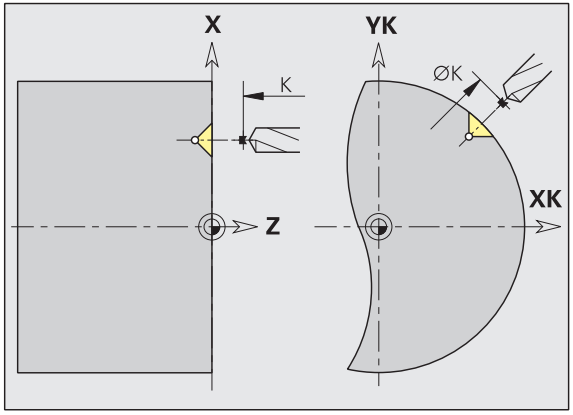
NS: Contour block number with boring geometry (G49, G300- or G310-Geo)

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

D: Retraction speed – default: 0
- D=0: Rapid traverse
- D=1: Feed rate

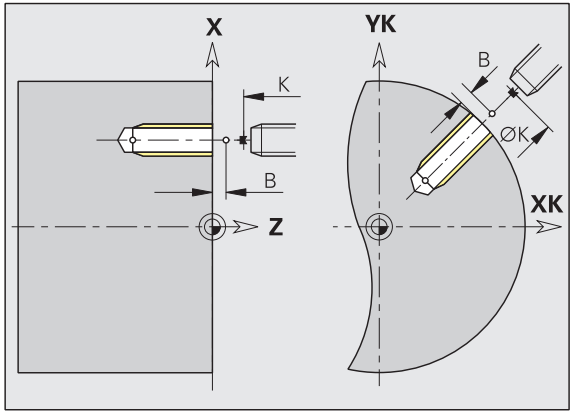
K: Retraction level (radial hole: diameter) – no input: tool returns to starting position/moves to clearance height

Hole pattern: "NS" points to the contour of the hole (not to the pattern definition).



Tapping G73

G73 can be used for axial and radial threads using driven or stationary tools.



Parameters

NS: Contour block number with boring geometry (G49, G300- or G310-Geo)

B: Chamfer length – default: Machining Parameter 7 "Thread chamfer length [GAL]"

S: Retraction speed – default: tapping speed

K: Retraction level (radial holes and holes in the YZ plane: Diameter) – No input: Tool moves to starting position or safety clearance

J: Pull-out length during use of collets with length compensation – default: 0

- Hole pattern: "NS" refers to the bore hole contour (and not the definition of the pattern).
- "Cycle STOP" only becomes effective at the end of a thread cut.
- Feed rate override is not effective.
- Do not use spindle override !

Deep drilling G74

G74 can be used for axial and radial holes on the end faces or lateral surface using driven or stationary tools. The hole is drilled in several passes.

The cycle is used for:

- A single hole without contour description
- Boring with contour description (single hole or hole pattern)

Parameters

NS: Contour block number with boring geometry (G49, G300- or G310-Geo) – no input: single hole without contour description

X, Z: Position, length – end points of axial/radial holes (X diameter)

P: 1st drilling depth

I: Reduction value – default: 0

B: Return distance
default: Retract to "starting point of hole"

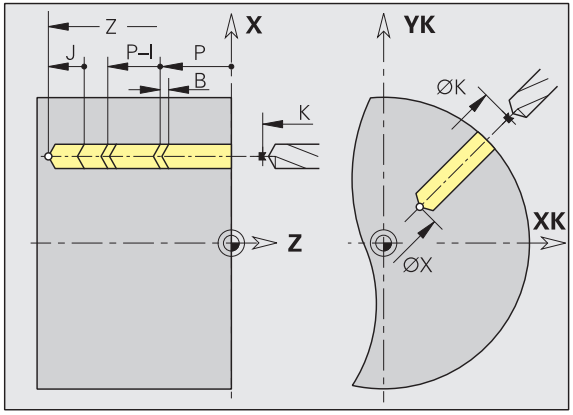
J: Minimum hole depth - default: 1/10 of P

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

V: Feed rate reduction (50%) – default: 0
- V=0 or 2: Feed rate reduction at start
- V=1 or 3: Feed rate reduction at start and end
- V=4: Feed rate reduction at end
- V=5: No feed rate reduction

Exception for V=0 and V=1: No feed rate reduction when boring/drilling with indexable inserts and twist drills with 180° angle

D: Retraction speed and infeed within the hole – default: 0
- D=0: Rapid traverse
- D=1: Feed rate



K: Retraction level (radial holes: diameter) – no input: tool returns to starting position/moves to clearance height

- Single hole without contour description: Program „X or Z" as alternative.
- Hole with contour description: Do not program „X, Z".
- Hole pattern: "NS" refers to the bore hole contour (and not the definition of the pattern).
- A "feed rate reduction at end" goes into effect only at the last drilling stage.

Reference diameter G120

G120 determines the reference diameter of the unrolled cylindrical surface. G120 is necessary if you are using "CY" (with G110...G113). G120 is a modular function.

Parameters
X: Diameter

Datum shift in C axis (absolute) G152

G152 defines a datum shift for the C axis (reference: Machine Parameter 1005, ff "Reference point C axis"). The datum is valid until the end of the program.

Parameters
C: Angle of the new C-axis datum

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 used only for cylindrical surface machining. An automatic modulo 360° function is carried out on the end faces.

Rapid traverse on end face G100

The tool moves at rapid traverse along the shortest path to the "end point."

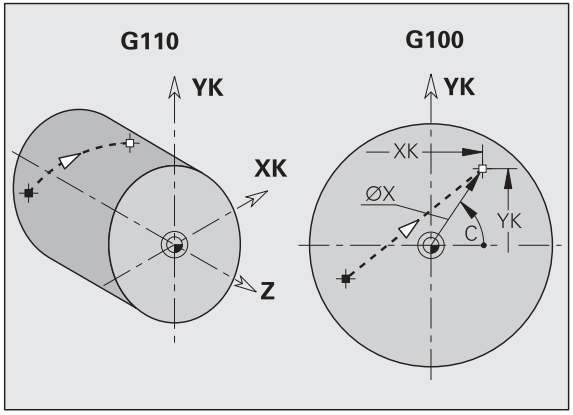
Parameters

X, C: End point, end angle

XK,YK: End point

Z: end points

Program either X, C or XK, YK.



Linear segment on end face G101

The tool moves at the programmed feed rate on a line to the "end point."

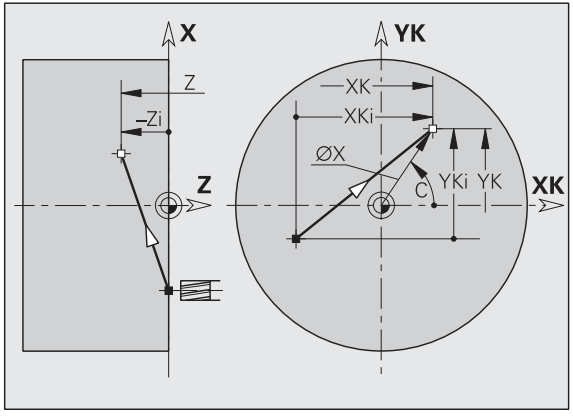
Parameters

X, C: End point, end angle

XK,YK: End point

Z: End depth – default: current Z position

Program either X, C or XK, YK.



Circular arc on end face G102 / G103

The tool moves in a circular arc at the feed rate to the "end point."

Parameters

X, C: End point, end angle

XK,YK: End point

R: Radius

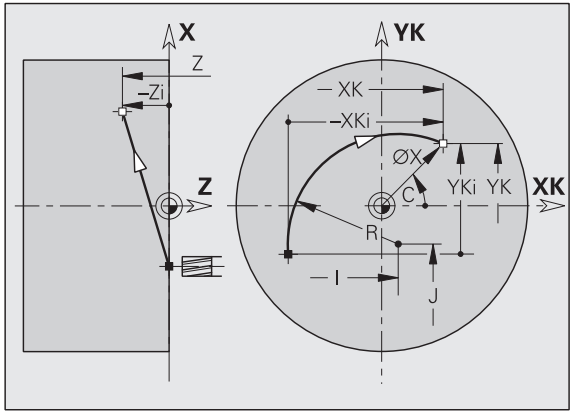
I, J: Center point (in Cartesian coordinates)

Z: End depth – default: current Z position

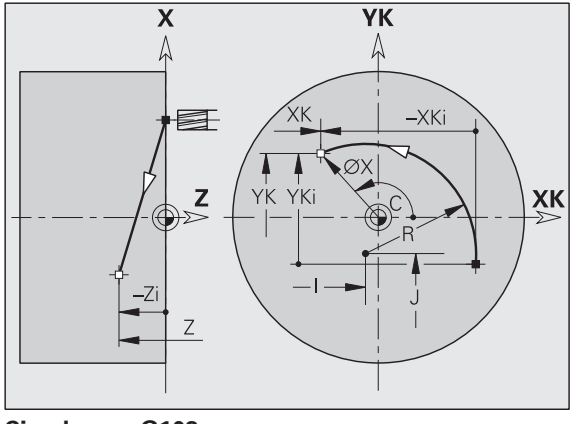
H: Circular plane (working plane) – default:
- H=0, 1: normal facing (XY plane)
- H=2: Machining in YZ plane
- H=3: Machining in XZ plane

K: Midpoint (Z-direction) – only for H=2, 3

- Program either X, C or XK, YK.
- Program either center or radius.
- Radius input: circular arc only possible ≤ 180°
- If the end point is the original coordinate starting point, enter XK=0 and YK=0.



Circular arc G102



Circular arc G103

Rapid traverse on cylindrical surface G110

The tool moves at rapid traverse along the shortest path to the "end point."

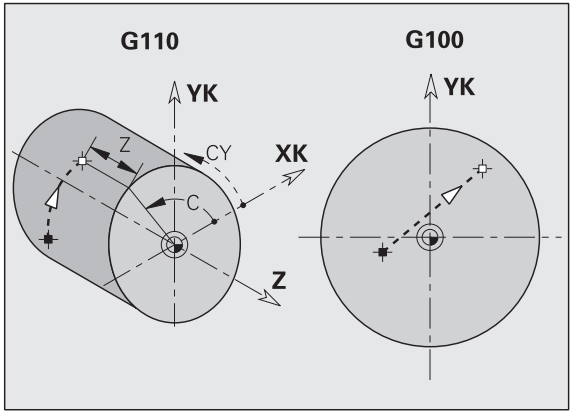
Parameters

Z, C: End point, end angle

CY: End point as linear value (referenced to unrolled reference diameter G120)

X: End point (diameter)

Program either Z, C or Z, CY.



Linear movement on the cylindrical surface G111

The tool moves at the programmed feed rate on a line to the "end point."

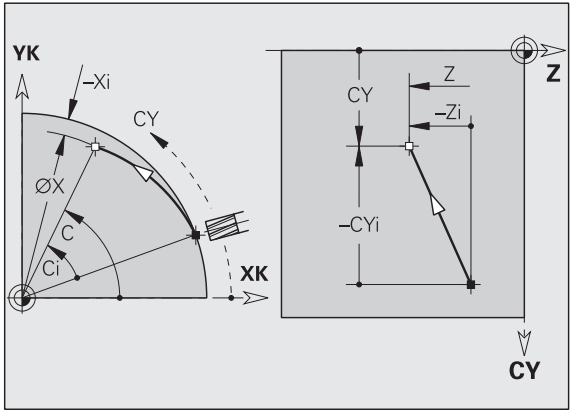
Parameters

Z, C: End point, end angle

CY: End point as linear value (referenced to unrolled reference diameter G120)

X: End depth (diameter value) – default: current X-position

Program either Z, C or Z, CY.



Circular arc on lateral surface G112 / G113

The tool moves in a circular arc at the feed rate to the "end point."

Parameters

Z, C: End point, end angle

CY: End point as linear value (referenced to unrolled reference diameter G120)

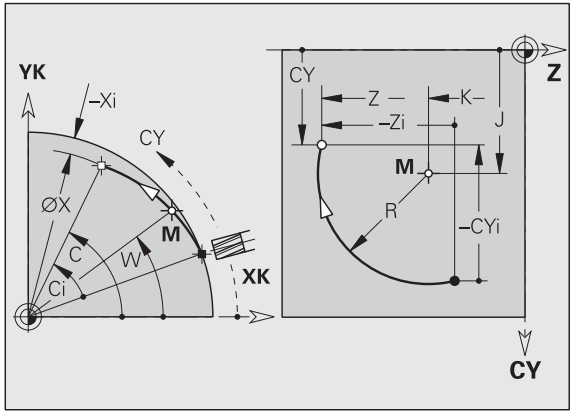
R: Radius

K, W: Position, angle to midpoint

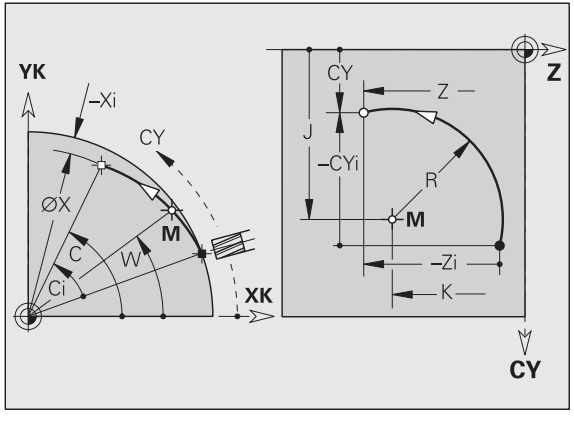
J: Center point coordinate as a linear value (referenced to unrolled G120 reference diameter)

X: End depth (diameter value) – default: current X-position

- Program either Z, C or Z and CY, or K, W or K and J.
- Program either center or radius.
- If you program a radius, the arc size cannot exceed 180°.



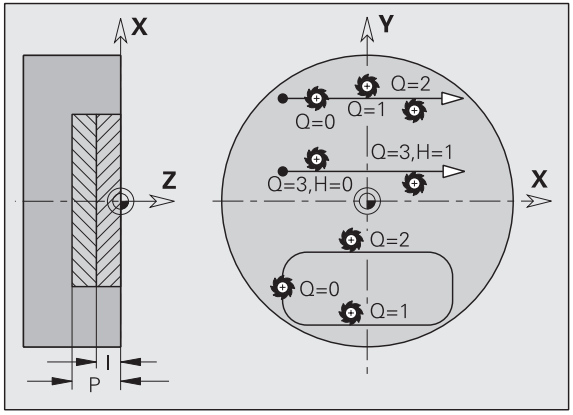
Circular arc G112



Circular arc G113

Contour milling G840

G840 mills, finishes, engraves or deburrs figures or "free contours" (open or closed).



Parameters

Q: Cycle type (= milling location)
- Q=0: milling center on the contour (without radius compensation)
- 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 (only with open contours): It depends on the "cutting direction H" **and** the direction of tool rotation whether the tool cuts from the left or right of the contour.

NS: Block number – Beginning of contour section
- Figures: Block number of the Figure
- "free contour" (open or closed): first contour element (not starting point).

NE: Block number – End of contour section
- Figures: no entry
- Closed contours: entry unnecessary
- Open contours: last contour element to be milled
- Contour consists of one element: entry unnecessary

H: Cutting direction – default: 0
- H=0: up-cut milling
- H=1: climb milling

I: (Maximum) infeed – no entry: milling in one infeed

F: Feed rate for infeed – default: active feed rate

E: Reduced feed rate for circular elements – no entry: current feed rate

R: Radius of approaching/departing arc – default: 0
- R=0: contour element is approached directly; feed to starting point above the milling plane – then vertical plunge
- R>0: Tool moves on an approaching/departing arc that connects tangentially to the contour element.
- R<0 for inside corners: tool moves on an 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.

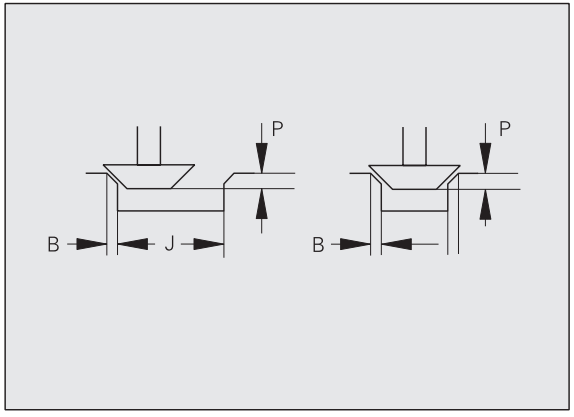
P: Milling depth
- Milling, finishing – no entry: milling depth from the contour description
- Deburring: plunging depth of the tool

K: Return plane – no input: Tool returns to the starting position
- Front or rear face: Return position in Z direction
- lateral surface: Return position in X direction (diameter)

B: Chamfer width for deburring the upper edges (sign has no significance)

J: Preparation diameter (tool diameter from machining)
- Required for deburring of open contours.
- Not required, if diameter of deburring tool = diameter of milling tool.

D, V: Beginning, end of element number for figures (only if partial figures are machined)




See the "CNC PILOT 4290 User's Manual" for the figure element numbers.

Pocket milling – roughing G845
Pocket milling – finishing G846

G845 roughs, G846 finishes closed contours and figures.

Parameters

NS: Block number (reference to contour section)

P: (Maximum) milling depth (infeed in the working plane)

R: Radius of approaching/departing arc – default: 0 (only with G846)
- R=0: contour element is approached directly; feed to approach point above the milling plane – then perpendicular feed to plunging depth
- R>0: mill moves along approaching/departing arc, which connects tangentially with the contour element

I, K: Allowance in X, Z direction (only with G845)

U: (minimum) overlap factor – overlap of tool paths (overlap = U* cutter diameter) – default: 0.5

V: Overrun factor – has no significance for machining with the C axis

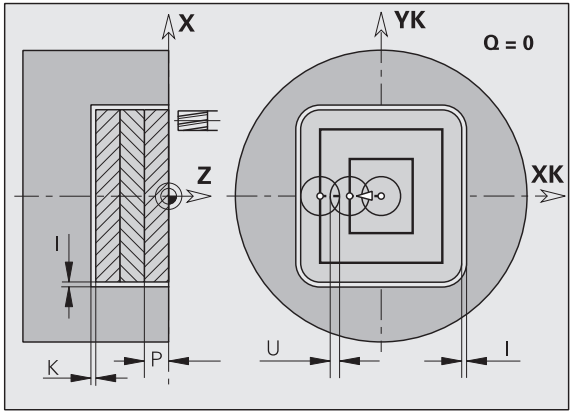
H: Cutting direction – default: 0
- H=0: up-cut milling
- H=1: climb milling

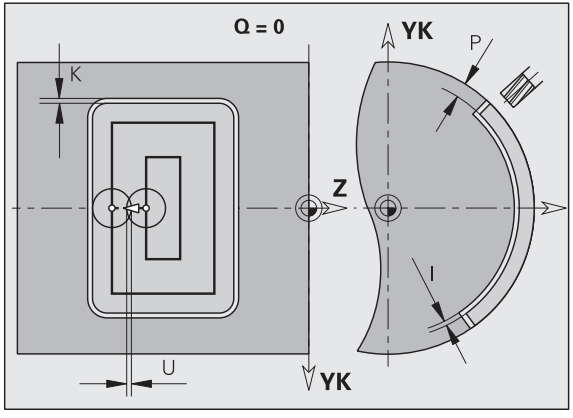
F: Feed rate for infeed – default: active feed rate

E: Reduced feed rate for circular elements – no entry: current feed rate

J: Return plane – no input: Tool returns to the starting position
- Front or rear face: Return position in Z direction
- lateral surface: Return position in X direction (diameter)

Q: Machining direction – default: 0
- Q=0: from the inside toward the outside
- Q=1: from the outside toward the inside





Period of dwell G4

The CNC PILOT interrupts the program run for the programmed length of time 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.

Parameters

F: Period of dwell
Range: 0 sec < F < 99.999 sec

Precision stop ON G7

G7 switches the "precision stop" on modally. With a precision stop the CNC PILOT does not run the following block until the last point has been reached in the tolerance window for position (tolerance window: machine parameter 1106, ff "position control for linear axis").

A 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 the precision stop off. The block containing G8 is executed **without** a precision stop.

Block precision stop G9

G9 activates a precision stop for the NC block in which it is programmed (see also "G7").

Switch off protection zone G60

resets protection zone monitoring. G60 is programmed **before** the traversing command that is to be monitored or not monitored.

Parameters

Q:
- No input: Deactivate protective zone monitoring for this block
- Q=0: Activate protective zone (modal)
- Q=1: Deactivate protective zone (modal)

Chuck selection G65

G65 shows the chuck in the simulation. G65 must be programmed separately for every chuck. G65 H.. without X, Z deletes the chuck.

Chucks are described in the database and are defined in CLAMPING DEVICE (H=1..3).

"Q" determines the degree and the type of clamping (external or internal).

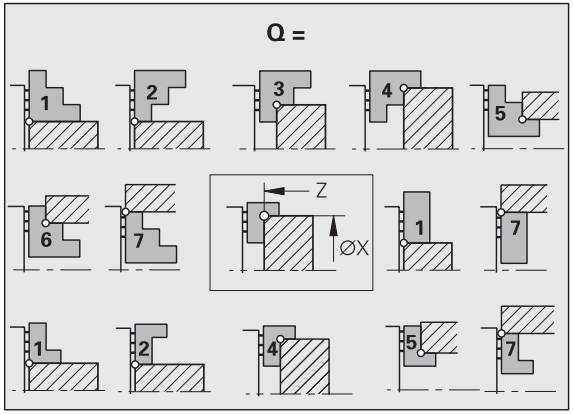
Parameters

H: Chuck number (H=1..3: Reference to CLAMPING DEVICE)

X, Z: Starting point – Position of the chuck reference point (X diameter) – **Reference: workpiece datum**

D: Spindle number (reference: "CLAMPING DEVICE" section)

Q: Grip form for chuck jaws (H=2) – no input: Q from the "CLAMPING DEVICE" section.



Spindle with workpiece G98

With G98 you can define which spindle is holding the workpiece.

The assignment of workpiece to spindle is necessary for thread cutting, boring and threading cycles if the workpiece is not being held by the main spindle.

Parameters

Q: Spindle number – default: 0 (main spindle)

Storing/loading contour follow-up G702

G702 Q0 stores the current contour without influencing the contour follow-up function.

G702 Q1 loads the stored contour; follow-up continues with the new contour.

Parameters

Q: Store/load current contour
- Q=0: store
- Q=1: load

Spindle synchronization G720

G720 controls the workpiece transfer from the master to the slave spindle and synchronizes functions such as polygonal turning jobs.

Where: **Q * master speed = F * slave speed**

Parameters

S: Number of the master spindle [1..4]

H: Number of the slave spindle [1..4] – no input or H=0: Switches off the spindle synchronization

C: Offset angle [°] – default: 0°

Q: Master speed factor – default: 1; Range: –100 ≤ Q ≤ 100

F: Slave speed factor – default: Q take effect; Range: –100 ≤ F ≤ 100

Regeneration of the contour G703

If changeable variables are used during run-time conditions "{V...}," CNC PILOT switches off the contour regeneration function within an IF program branch and during a WHILE loop. It is switched on again after ENDIF/ENDWHILE.

With G703 you can switch on the contour regeneration function for THEN or ELSE branches.

Parameters

Q: Contour regeneration On/Off
- Q=0: Off
- Q=1: On

C-angle offset G905

G905 measures the angular offset which occurs during workpiece transfer with a "rotating spindle". The programmed "angle C" is added to the measured offset and becomes effective as the datum shift value for the C axis.

Parameters

Q: Number of the C axis

C: Angle of additional datum shift for displaced infeed – Range: –360° ≤ C ≤ 360°; default: 0°

Look-ahead G918

Look-ahead control can be switched On/Off with G918. G918 is programmed in a separate NC block before/after thread machining (G31, G32, G33).

Parameters

Q: Look-ahead On/Off – default: 1
- Q=0: off
- Q=1: on

Spindle override 100% G919

The spindle override function can be switched On/Off with G919.

Parameters

Q: Spindle number – default: 0

H: Type of limit – default: 0
- H=0: switch on spindle override
- H=1: switch spindle override to 100% – modular
- H=2: switch spindle override to 100% – for the current NC block

Deactivating zero point shifts G920

G920 deactivates the workpiece zero point and all zeto point shifts. Traverse paths and position values are referenced to the distance **tool tip – machine datum**.

Zero point shifts, deactivating tool lengths G921

G921 "deactivates" the workpiece datum, all datum shifts and the tool dimensions. Traverse paths and position values are referenced to the distance **slide reference point – machine datum.**

Activate zero point shifts G980

G980 activates the workpiece datum and all datum shifts.

Traverse paths and position values are now referenced to the distance **tool tip – workpiece datum**, while taking the datum shifts into consideration.

Zero point shifts, activating tool lengths G981

G981 activates the workpiece datum, all datum shifts and the tool dimensions.

Traverse paths and position values are now referenced to the distance **tool tip – workpiece datum**, while taking the datum shifts into consideration.

Subprograms

Calling a subprogram: L"12345678" V1

Meaning:

- External subprograms are defined in a separate file. They can be called from any main program, other subprograms, or from TURN PLUS.
- Local subprograms are in the same file as the main program. They can only be called 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 20 „transfer values" to a subprogram. The designations (parameter designations) are:
  LA..LF, LH, I, J, K, O, P, R, S, U, W, X, Y, Z.
  The transfer values are available as variables within the subprogram. The identification code is: „#__.", followed by the parameter designation in lower-case letters (for example: #__la).
  You can use the transfer values when programming with variables within the subprogram.
- The variables #256..#285 are available in every subprogram as local variables.
- If a subprogram is to be executed repeatedly, enter the number of times the subprogram is to be repeated in the parameter "number repeats Q."
- 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.