DOCUMENTATION ISG-kernel

Functional description
Turning

Short description:
FCT-S3

Documentation version: 1.09
04/06/2025

Preface

Legal information

This documentation was produced with utmost care. The products and scope of functions described are under continuous development. We reserve the right to revise and amend the documentation at any time and without prior notice.

No claims may be made for products which have already been delivered if such claims are based on the specifications, figures and descriptions contained in this documentation.

Personnel qualifications

This description is solely intended for skilled technicians who were trained in control, automation and drive systems and who are familiar with the applicable standards, the relevant documentation and the machining application.

It is absolutely vital to refer to this documentation, the instructions below and the explanations to carry out installation and commissioning work. Skilled technicians are under the obligation to use the documentation duly published for every installation and commissioning operation.

Skilled technicians must ensure that the application or use of the products described fulfil all safety requirements including all applicable laws, regulations, provisions and standards.

Further information

Links below (DE)

https://www.isg-stuttgart.de/produkte/softwareprodukte/isg-kernel/dokumente-und-downloads

or (EN)

https://www.isg-stuttgart.de/en/products/softwareproducts/isg-kernel/documents-and-downloads

contains further information on messages generated in the NC kernel, online help, PLC libraries, tools, etc. in addition to the current documentation.

Disclaimer

It is forbidden to make any changes to the software configuration which are not contained in the options described in this documentation.

Trade marks and patents

ISG®, ISG kernel®, ISG virtuos®, ISG dirigent®, TwinStore® and the associated logos are registered and licensed trade marks of ISG Industrielle Steuerungstechnik GmbH.

The use of other trade marks or logos contained in this documentation by third parties may result in a violation of the rights of the respective trade mark owners.

Copyright

General and safety instructions

Icons used and their meanings

This documentation uses the following icons next to the safety instruction and the associated text. Please read the (safety) instructions carefully and comply with them at all times.

Icons in explanatory text

➢ Indicates an action.

⇨ Indicates an action statement.

⚠ DANGER

Acute danger to life!

If you fail to comply with the safety instruction next to this icon, there is immediate danger to human life and health.

⚠ CAUTION

Personal injury and damage to machines!

If you fail to comply with the safety instruction next to this icon, it may result in personal injury or damage to machines.

Attention

Restriction or error

This icon describes restrictions or warns of errors.

Notice

Tips and other notes

This icon indicates information to assist in general understanding or to provide additional information.

Example

General example

Example that clarifies the text.

Programing Example

NC programming example

Programming example (complete NC program or program sequence) of the described function or NC command.

Release Note

Specific version information

Optional or restricted function. The availability of this function depends on the configuration and the scope of the version.

Contents

List of figures

1 Overview

Task

The CNC provides specific functions for turning work.

Possible applications

The following functions are available:

- Diameter programming
- Tool tip radius compensation (SRK)
- Thread cutting with endlessly rotating spindle (G33)
- Thread tapping without compensation chuck (G63, G331/G332)
- Feedrate per revolution (G95)
- Constant cutting speed (G96, G97, G196)
- Face and lateral surface machining (#CAX, #FACE, #CYL)

Programming

Refer to the appropriate sections for turning functions.

Parametrisation

Turning functions are configured by specific parameters which are described in the chapter Parameters [▸ 50].

Links to other documents

For the sake of clarity, links to other documents and parameters are abbreviated, e.g. [PROG] for the Programming Manual or P-AXIS-00001 for an axis parameter.

For technical reasons, these links only function in the Online Help (HTML5, CHM) but not in pdf files since pdfs do not support cross-linking.

2 Description

Turning is a manufacturing process in metal cutting and is regarded as one of the basic processes in mechanical metal cutting in addition to milling, drilling and grinding. The machine tool required for machining is the lathe. In its basic state, it consists of various components:

- a drive which rotates the workpiece fixed in the chuck
- a tool slide which bears the turning tool with handwheels and/or other drives to move the tool axes
- a stable and vibration-damping frame
- additional components, so-called bezels and a tailstock to support long turned parts

The simple mechanical shape of a universal lathe is depicted in the figure below. It is mainly used to produce small batches and manufacture one-off parts in workshops. For medium-size and large batch production, the so-called CNC lathe is used. It has a CNC controller which executes a programmed tool motion on complex paths.

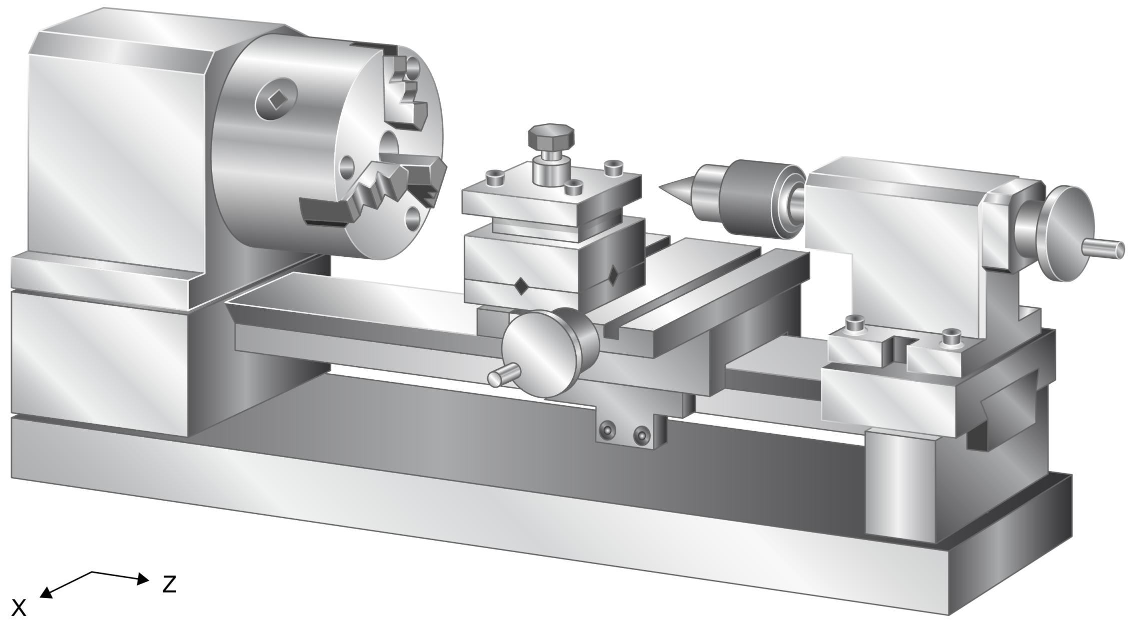


Fig. 1: Lathe

2.1 Standard turning functions

Standard turning functions are used for conventional turning work with an endlessly rotating spindle. A position-controlled interpolated CNC spindle axis must be configured as the main spindle to achieve the full scope of functions. For details see [FCT-S1].

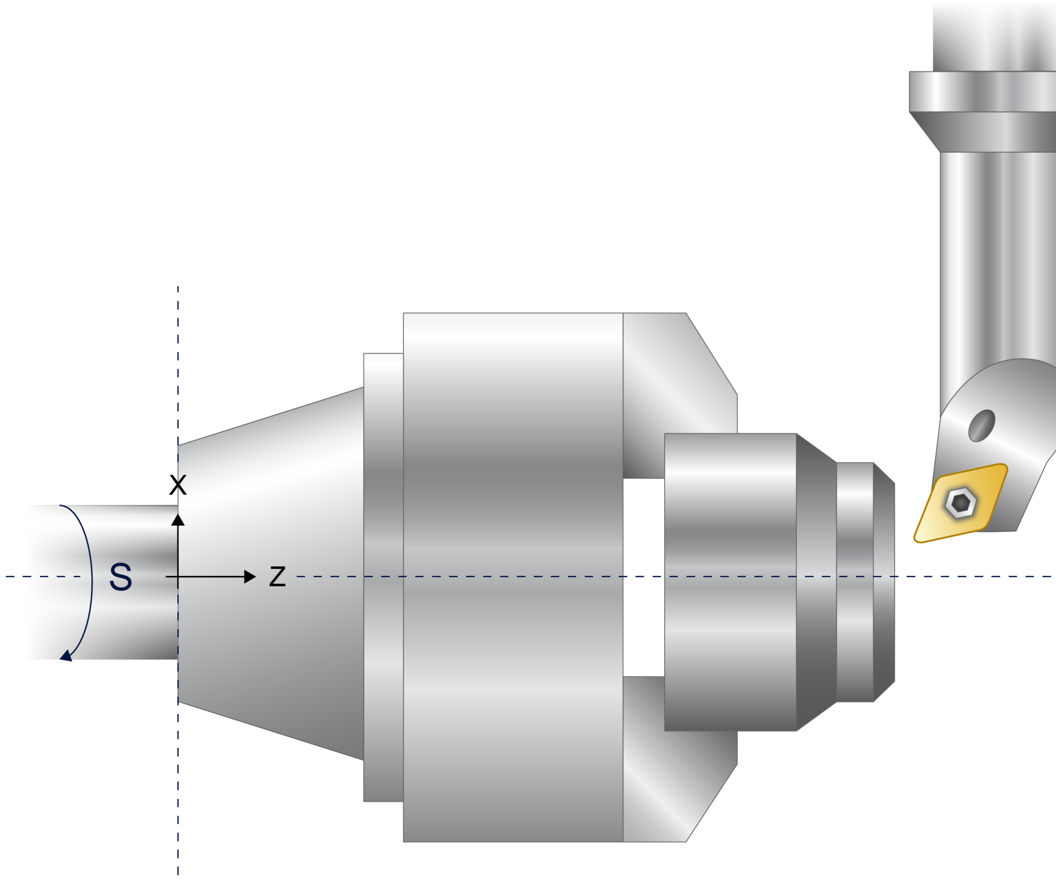


Fig. 2: Turning work

The tool-carrying spindle axis is commanded at the assigned speed by the spindle commands M03, M04 and M05.

A longitudinal and a face turning axis must be identified by an entry in the axis parameter P-AXIS-00015. The figure above shows an example where the Z axis is marked as the longitudinal turning axis (bit 0x80) and the X axis as the face turning axis (bit 0x40).

The conventional axis configuration consists of 3 Cartesian main axes in the order X-Y-Z (1st main axis, 2nd main axis, 3rd main axis). Machining is executed in the programmed G18 plane (Z, X).

Notice

Recommendation: Configuration of the main axes X-Y-Z and machining in the G18 plane.

Machining can also be executed in the default G17 plane. The alternative axis configuration must then be Z-X-Y.

Standard turning functions

Function	Meaning	Unit
G51	Diameter programming	-
G52	Deselection of diameter programming	-
G41, G42	Tool tip radius compensation, left/right	-
G40	Deselect tool tip compensation	-
G95	Feed rate per revolution	**F [mm/rev]**
G33	Thread cutting	**K [mm/rev]**
G96	Constant cutting speed	**S [m/min]**
G196	Max. speed for G96	**S [rpm]**
G94	Default unit for path feed	**F [mm/min]**
G97	Default unit for spindle speed	**S [rpm]**
F: Path feed S: Spindle speed, cutting speed K: Pitch		

2.1.1 Diameter programming (G51/G52)

Syntax:

G51	Select diameter value	modal
G52	Deselect diameter value	modal, initial state

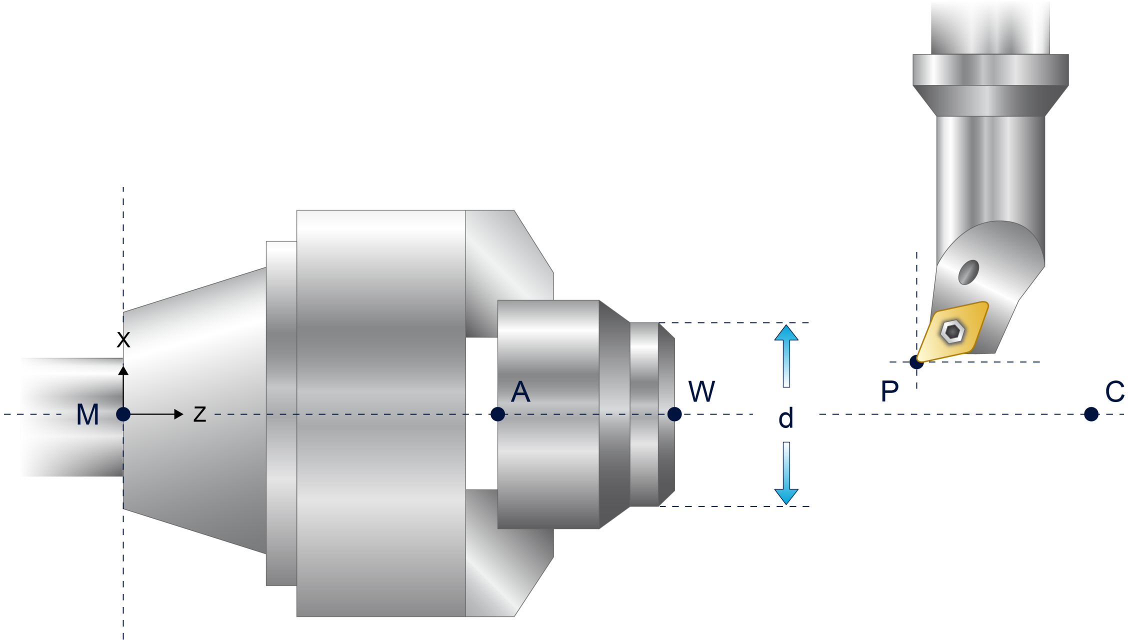


Fig. 3: Reference points and diameter programming

W Workpiece zero point
X Face turning axis
M Machine zero point
C Control zero point
P Cutting point
Z Longitudinal turning axis
A Fixed stop point
d Programmed dimension for diameter programming

When diameter programming is selected, the positional values for the face turning axis in a motion block are interpreted as diameter values relative to the turning centre point.

It should be noted that the programmed coordinates of the face turning axis only correspond to the workpiece diameter if the zero point of the face turning axis is located at the turning centre point (irrespective of whether offsets act as a diameter; see Programming example).

The axis parameters can parameterise axes in "face turning" mode:

- G51 with absolute programming (G90) (P-AXIS-00058)
- … and/or G51 with relative programming (G91) (P-AXIS-00059
- Reference point programming (G92) and zero point programming (G53 – G59) in the diameter (P-CHAN-00091)

G51 acts on the axis which is operated in "face turning" mode. When the face turning axis is selected, the face turning axis must exist in the machining plane (G17, G18, G19).

The coordinates of the circle centre point (I, J, K) and circle radius programming (R) are not programmed in the diameter.

Diameter programming is deselected with G52.

Programing Example

Diameter programming (G51/G52) in G18

```
;General settings (optional):
;Display position values in the diameter P-CHAN-00256 (TRUE, 1)

;Settings of X axis:
;Face turning axis, translatory: P-AXIS-00015 (0x41)
;G51 with G90: P-AXIS-00058 (TRUE, 1)
;G51 with G91: P-AXIS-00059 (FALSE, 0)(optional)
;G92, G53-G59 in the diameter: P-CHAN-00091 (TRUE, 1) (optional)

;Settings of Z axis:
;Longitudinal turning axis, translatory: P-AXIS-00015 (0x81)
;
N05 G18
N10 G90 G01 F1000
N20 G51 X80         ;diameter 80 mm
N30 G92 X10         ;G92 by 10 mm in the diameter
N40 X0              ;position 0 + G92 => diameter 10 mm
N50 G91 X50         ;X relative +50mm, in the diameter
N80 G52             ;deselect diameter programming
;…

N90 M30
```

2.1.2 Cutter radius compensation (G40/G41/G42)

`Syntax:`

G40	Deselect CRC	modal, initial state
G41	CRC left of contour	modal
G42	CRC right of contour	modal

Tool tip radius compensation (SRK) acts in the machining plane selected using G17, G18, G19 for turning work. In this plane, one of the axes must be operated in "face turning" mode and the other in "longitudinal turning" mode. (axis mode: P-AXIS-00015)

The data records stored in the D words are used as tool compensation values. For turning tools, the orientation of the cutter edge relative to the machining plane (face/longitudinal turning axis) must be specified in the parameter P-TOOL-00002 by an additional identifier 1...9 (see figure).

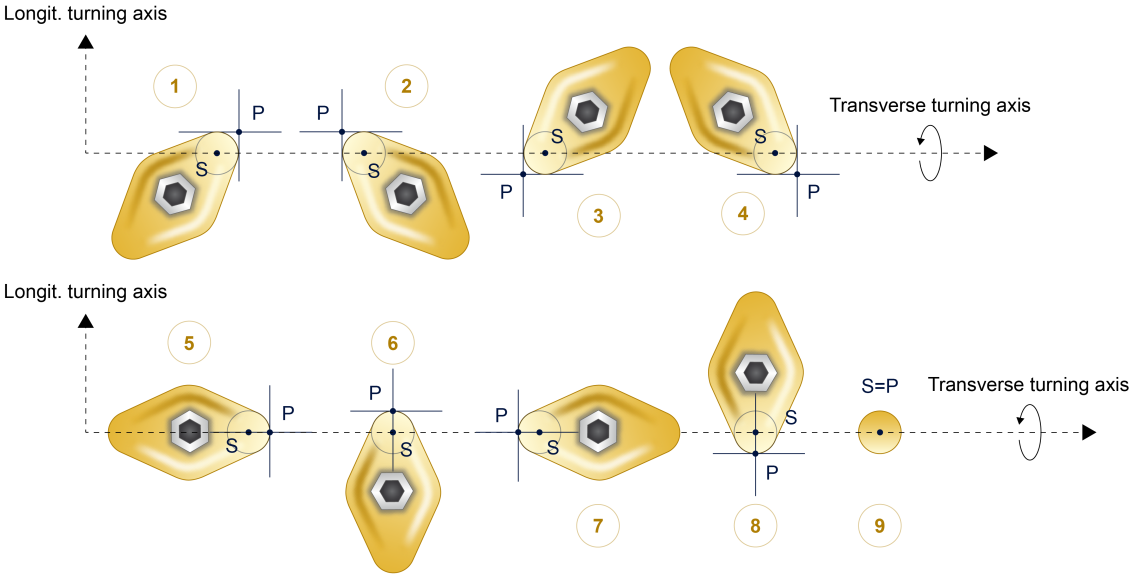


Fig. 4: Orientation of cutter edge to machining plane.

A typical turning tool is characterised by the following values/parameters:

• Tool type	1 (turning tool)
• SRK orientation	1...9
• Tool radius	Tool tip radius
• Tool length	--
• Tool offset (see figure below)	

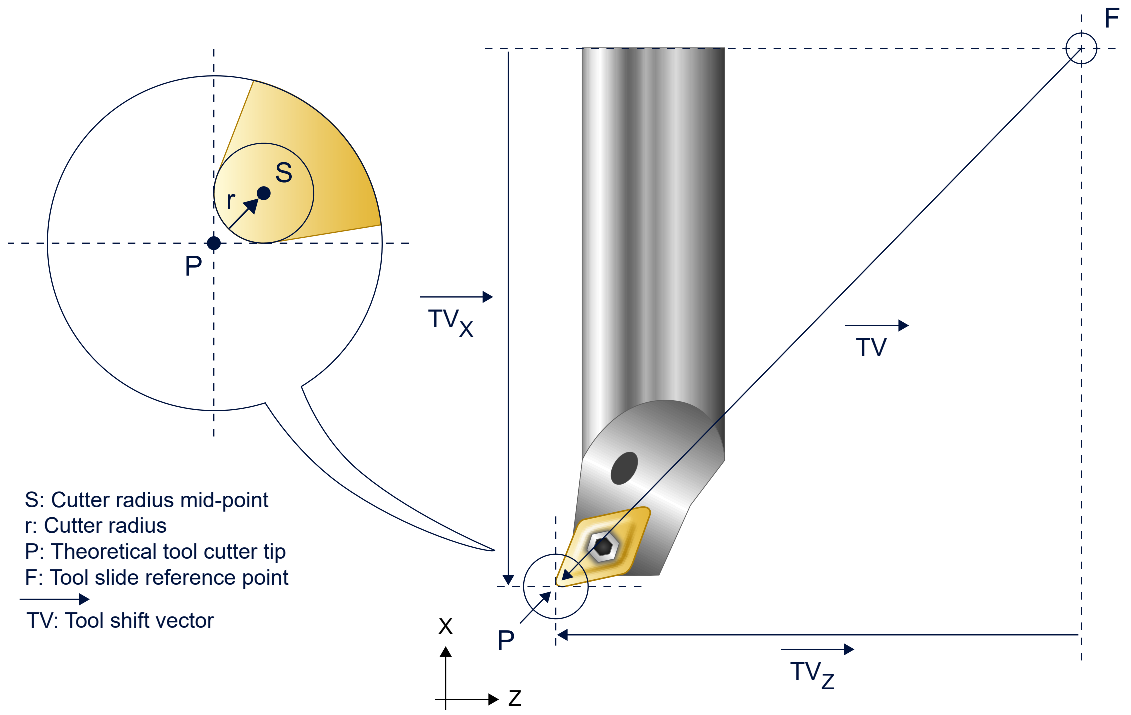


Fig. 5: Tool gauging for tool offset compensation.

When specifying tool axis offsets, their mathematical sign should be noted since it refers to components of the tool offset vector in the machining plane. In the example of a turning tool shown in the above figure, the offsets in the direction of the X and Y axis both have negative (mathematical) signs.

Tool offsets must be specified up to the theoretical tool top (point P).

A change between turning tool and milling tool is permitted when G41 or G42 is selected. With absolute programming (G90) the current axis offset values of the new tool are included in the calculation for the next motion block depending on the tool type.

2.1.3 2D cutter radius compensation (CRC) and 5-axis kinematics

Release Note

This function is available as of CNC Build V3.1.3081.2 or V3.1.3108.1.

This function allows 5-axis machines to also be used for conventional turn machining work. The driven milling tool is replaced by a fixed turning tool with a cutting insert. The kinematic ID identifies the related transformation as an element of the data set of kinematic parameters. The second rotary axis in the kinematic has no function in this application. It is replaced by a simulated axis and remains set to position 0. The physical C axis of the machine is commanded as a spindle and rotates the workpiece in continuous operation. The axis can therefore be configured as a spindle axis, for example, and be exchanged in coordinated motion for milling operation.

The basic axis and channel parameters required for using cutter radius compensation are described in [FCT-S3].

For use with a 5-axis kinematic, the channel parameter P-CHAN-00456 must also be set as follows:

```
trafo_mode          TOOL_TIP_RADIUS_COMP_IN_PLANE
```

2D CRC can be used in combination with kinematics.

The following kinematics support this function:

- KIN_TYP_57 – 5-axis kinematics with B/C workpiece table
- KIN_TYP_58 – Five-axis kinematics with A/C workpiece table

The tool can be oriented in relation to the workpiece at the CRC level, enabling more complex contours to be machined on the facing and longitudinal turning sides using a tool cutter edge position.

The CNC calculates CRC offsets depending on the orientation angle of the cutter edge in relation to the workpiece; the cutter edge is correctly guided during orientation motions.

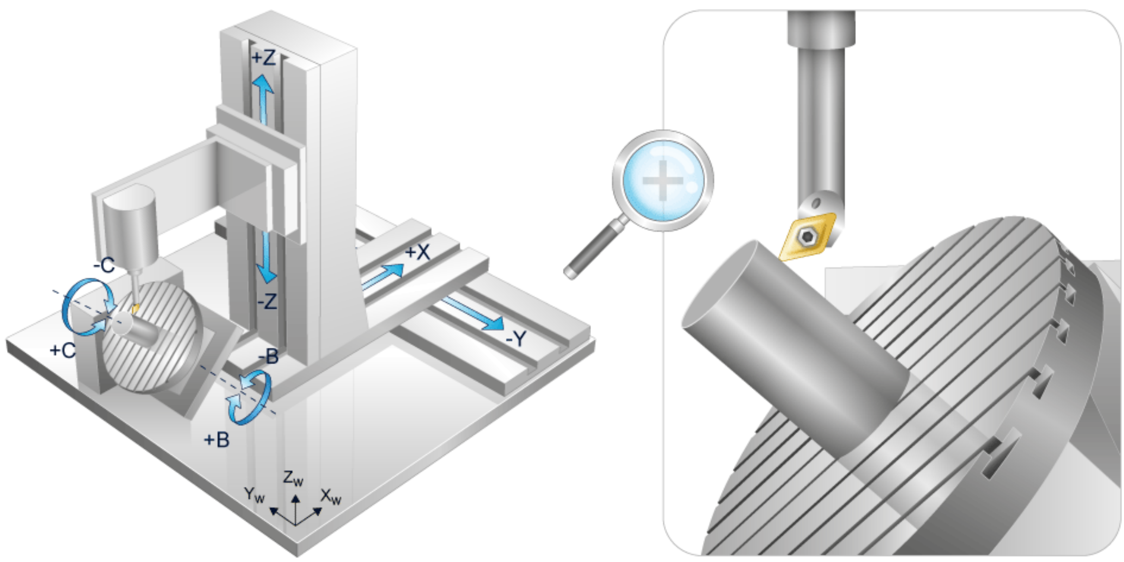
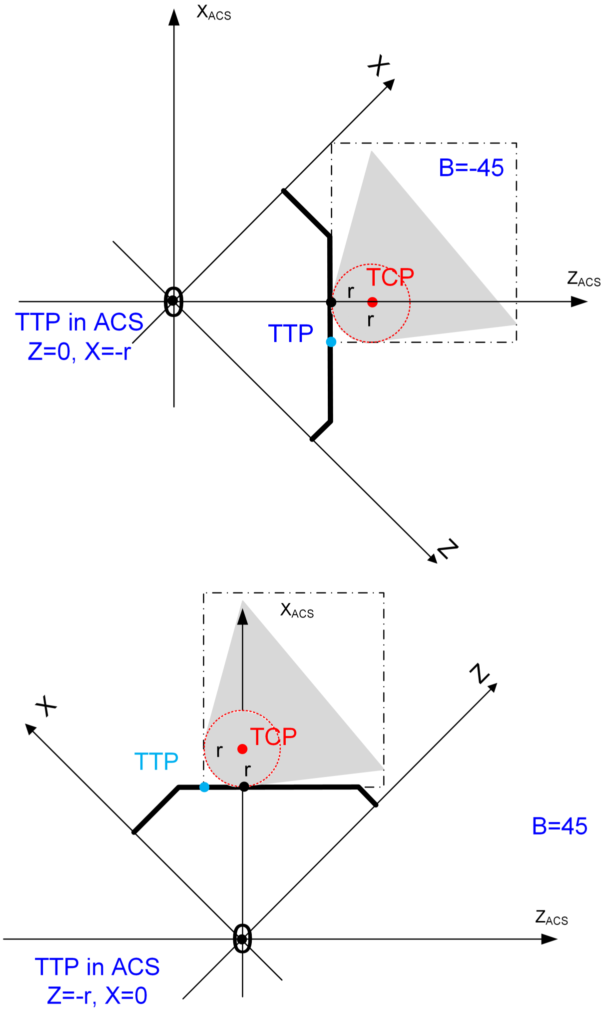


Fig. 6: Cutter radius compensation with kinematic 57

Position of the contact point depending on the orientation. The point designated by TTP denotes the tool tip point (**T**ool **T**ip **P**oint).



Each contact point is also dependent on the cutter edge position of the turning tool. See Cutter radius compensation (G40/G41/G42) [▸ 12]

Programing Example

Cutter radius compensation with kinematic 57

```
%L SUB_MV_CUTTER_RADIUS_COMP
N1100 G51
N1200 G18
N1300 G41
N1400 G01 G90 X100 Z30 F2000
N1500         X60  Z50
N1600              Z90    B45
N1700         X100 Z110
N1800              Z140
N1900                     B0
N2000         X60  Z160   B-45
N2100 G40
N2200 G90     X100 Z180   B0
M29

%main
N800 G01 G90 X0 Z200 F2000
N900 D1 ; -> CRC cutter edge position 3
N1000 M03 S500
N1100 #KIN ID[57]
N1200 #TRAFO ON
N1300 G90 B0
N1400   LL SUB_MV_CUTTER_RADIUS_COMP
N1500 #TRAFO OFF
M30
```

2.1.4 Feedrate per revolution (G95 / G94)

G95	Feedrate in mm/revolution	(modal)
G94	Feed in mm/min	(modal)

When turning with active G95, a constant chip thickness can be defined using the F word in mm/rev, irrespective of the spindle speed (rpm) and provided that the path dynamics act with a limiting effect.

The CNC calculates the current path feed from the current command spindle speed and the programmed feedrate per revolution.

Here, the axis feedrate is linked to the rotational speed (rpm) of the position-controlled spindle. It is only valid in conjunction with the G function with which it was programmed. Therefore, if there is a change from G95 to G94 or G93, the F word valid for G95 is not adopted.

Example: S: 1200 rpm, F: 1.5mm/rev -> VB = 1800mm/min

Programing Example

Feedrate per revolution

```
N10 F1000 X100 M3 S1200 ;feedrate 1000 mm/min G94

N20 G95 F1.5            ;feedrate 1.5 mm/rev, spindle speed 1200 rpm)

N30 G94 X50             ;feedrate 1000 mm/min valid from N10

N40 G93 F20 X20         ;machining time 20 s

N50 G95 Y200 S2000      ;feedrate 1.5 mm/rev valid from N20;
                        ;spindle speed 2000 1/min)

N60 M30
```

2.1.5 Thread cutting with endlessly rotating spindle (G33)

Single-start/multi-start threads

When thread cutting with an endlessly rotating spindle (G33), the path motion is synchronised to the zero passage of the spindle rotation. Therefore, the thread can also be cut in several passes in succession. When an offset angle is specified as option, multi-start threads can also be produced.

To achieve a good machining result and to minimise path errors, feedforward control can be selected for the spindle and for path axes.

Programming

```
Syntax example for ZX plane (longitudinal axis Z, feed axis X):
```

G33 Z.. K.. [<*spindle_name*>**.OFFSET=..**] modal

G33	Thread cutting with endlessly rotating spindle. The G33 function is modal. The next motion block with a modal block type (G00, G01, G02, G03, spline, polynomial) deselects thread cutting.
Z..	Target point ("thread length") in [mm, inch]
K..	The thread pitch is programmed with active thread cutting in the unit [mm/rev, inch/rev] without a mathematical sign using the address letters I, J and K. They are assigned to the X, Y and Z axes according to DIN 66025. The thread pitch is modal up to program end and should not be zero on when G33 is selected. The feed is not programmed using the F word but results from the spindle speed and the thread pitch. The pitch of longitudinal or tapered threads at an inclination angle less than 45° is specified by the address letter K if the Z axis is the longitudinal turning axis. With facing or tapered threads with a pitch greater than or equal to 45°, the pitch is specified by I if the X axis is used as the face turning axis, and by J if the Y axis is used. The figure below shows examples for specifying thread pitch using the address letters in the Z-X plane.
<*spindle_name*>.OFFSET=..	Thread offset angle in [°] in spindle modulo range. Only required as an option for multi-turn threads. The offset angle is modal up to program end. Spindle name according to P-CHAN-00053. The "=" character is optional.

Pitch values I, K with longitudinal thread

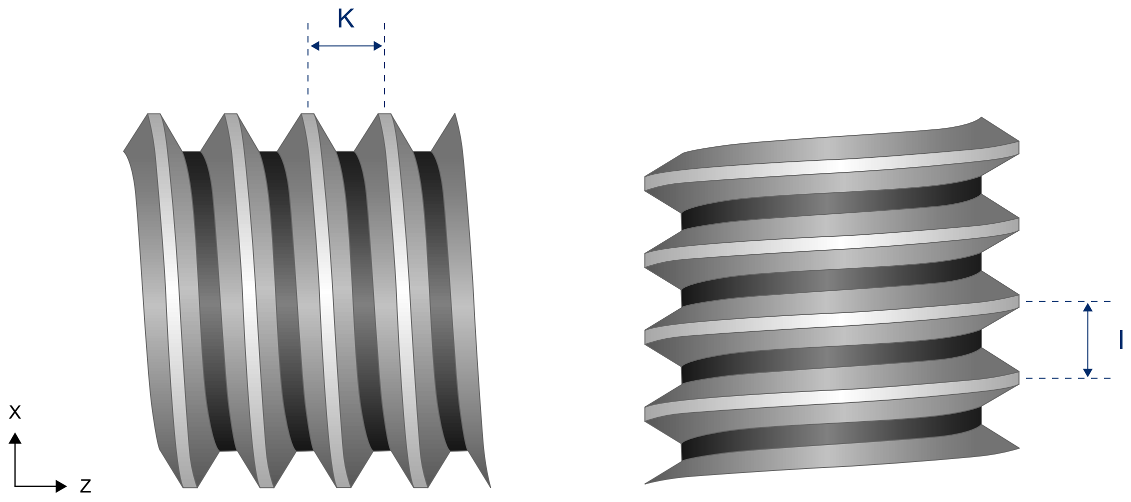


Fig. 7: Value of thread pitch for longitudinal thread

Pitch values I, K with tapered thread

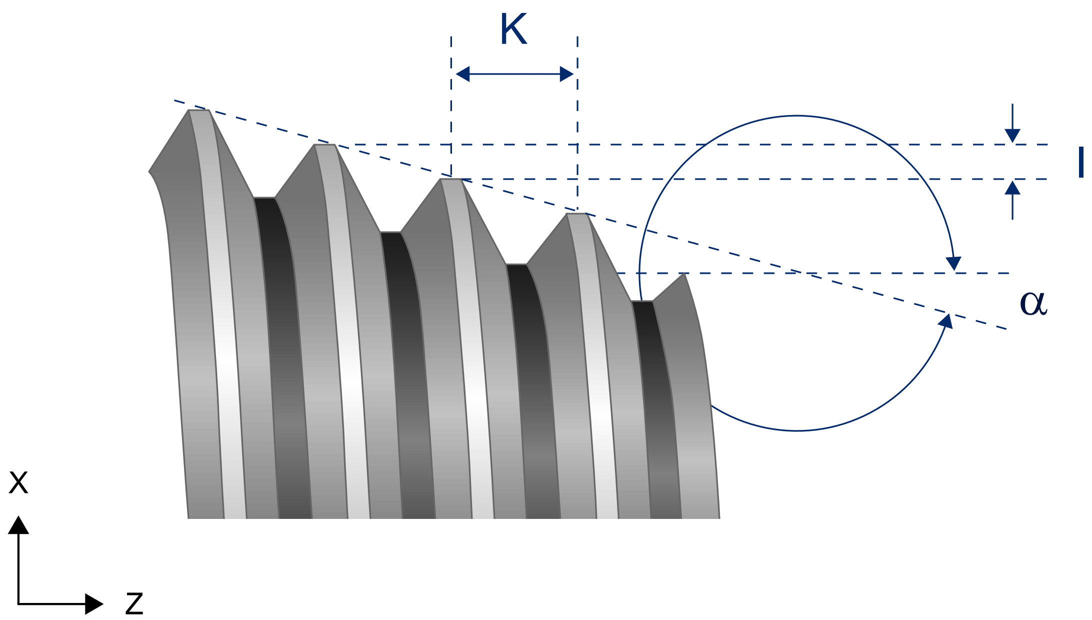


Fig. 8: Value of thread pitch for tapered thread

Programing Example

Thread cutting with endlessly rotating spindle (G33)

G33 Z.. K.. [S.OFFSET=..]

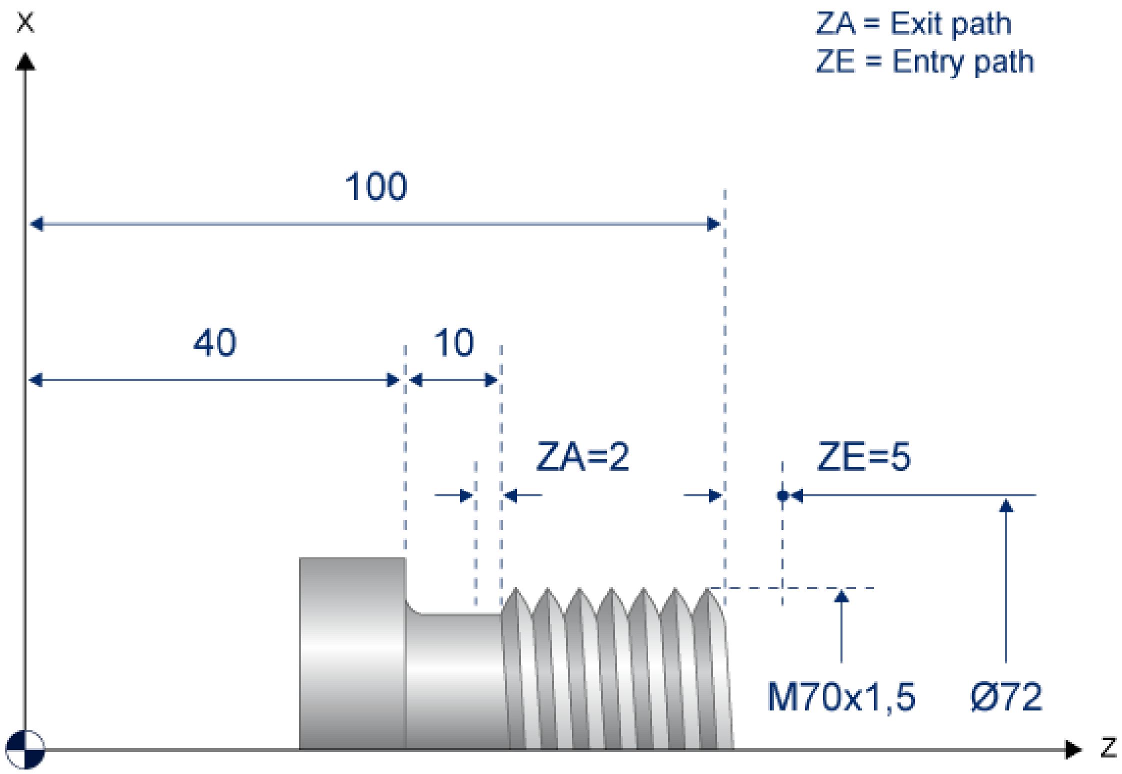


Fig. 9: Representation of geometry example

Cutting a longitudinal thread (M70x1.5) with several cuts:

```
%L longit_thread

N100 G33 Z48 K1.5                  ;Cut thread turn
N110 G00 X72                       ;Retract and move
N120 Z105                          ;to start position
N130 M29                           ;Subroutine end

%G33 (thread depth 0.92 mm)
N10 G51                            ;Select diameter programming
N15 T1 D1 M03 S400                 ;Select tool, start spindle
N20 G00 X72 Z105                   ;Approach

N25 G01 X69.54 F1000               ;Position at 1st cutting depth
N30 LL longitudinal thread               ;1st thread cut

N35 G01 X69.08                     ;Position at 2nd cutting depth
N30 LL longitudinal thread               ;2nd thread cut

N35 G01 X68.62                     ;Position at 3rd cutting depth
N30 LL longitudinal thread               ;3rd thread cut

N35 G01 X68.16                     ;Position at final depth
N30 LL longitudinal thread               ;4th thread cut

N35 G01 X68.16                     ;Reposition at final depth
N30 LL longitudinal thread         ;Empty cut

N60 M05 X150 Z200               ;Moving to end position
N65 M30                       ;Program end
```

Cut a 2-turn longitudinal thread (M70x1.5)

```
%G33_2 (2-turn thread, thread depth 0.92 mm)
N10 G51                            ;Select diameter programming
N15 T1 D1 M03 S40                 ;Select tool, start spindle
N20 G00 X72 Z105                   ;Approach
N25 G01 X68.16 F1000               ;Position at thread depth
N30 G33 Z48 K1.5                   ;Cut 1st thread turn
N35 G00 X72                       ;Retract and move
N40 Z105                           ;to next
N45 G01 X68.16                     ;start position
N50 G33 Z48 K1.5 S.OFFSET=180      ;Cut 2nd thread turn at 180°
N55 G00 X72                       ;Retract and move
N60 M05 X150 Z200               ;to end position
N65 M30                       ;Program end
```

Cutting a tapered thread

```
%L tapered thread
N010 G33 Z90 X1 I5.0                 ;Cut thread turn (reference I)
; N010 G33 Z90 X1 K5.0                ;Cut thread turn (reference K)
N020 G00 X72                      ;Retract and move
N030 Z105                         ;to start position
N040 M29                          ;Subroutine end

%G33
N050 G00 X0 Y0 Z0
N060 G18
N070 G51                           ;Select diameter programming
N080 D1 M03 S1                ;Select tool, start spindle
N090 G00 X105 Z105                  ;Start
N100 G01 X100 F1000              ;Position at 1st cutting depth
N110 LL tapered thread              ;1st thread cut
N120 M05 X150 Z200              ;Move to end position
N130 M30                      ;Program end
```

2.1.6 Thread cutting at actual spindle speed

When cutting a thread, the spindle speed may tend to deviate under load. In order to execute thread tapping in such cases, the motion of the linear axes can be directly coupled to the spindle's actual speed. This means that a path motion is also synchronised to the zero passage of the actual positions.

To avoid set value jumps in path axes, a very noisy actual spindle speed can be smoothed by an averaging filter.

Release Note

Function available as of V3.01.3080.16.

The function is activated either with Channel parameters [▸ 52] or with the NC command #TURN [▸ 36] [THREAD_CUT_ACT_SPEED=1 …].

Example

Thread cutting at actual spindle speed]

Define in channel parameters:

```
thread_cutting.use_actual_speed 1   ( P-CHAN-00834 [▸ 52])
thread_cutting.n_cycles         5   ( P-CHAN-00835 [▸ 53])
```

Alternative definition in NC program:

```
#TURN[THREAD_CUT_ACT_SPEED=1 THREAD_CUT_N_CYCLES=5]
```

Programing Example

Thread cutting at actual spindle speed

```
%spindle_test.nc
(Thread cutting at actual speed)
N15 M03 S40            ; Start spindle
N20 G00 X72 Z105       ; Approach
N25 G01 X68.16 F1000   ; Position at thread depth
N30 G33 Z48 K1.5       ; 1. Cut thread turn at command speed
N35 G00 X72            ; Retract
N40 Z105               ; and move
N45 G01 X68.16         ; to next start position

; Thread cutting at actual speed and filtered values
N50 #TURN[THREAD_CUT_ACT_SPEED=1 THREAD_CUT_N_CYCLES=20]
; 2. Cut thread turn at 180° at actual speed
N55 G33 Z10 K1.5 S.OFFSET=180

N60 G00 X72            ; Retract and move
N65 M05 X150 Z200      ; to end position
N70 M30
```

2.1.7 Thread chain

Before a thread cutting block, synchronisation normally takes place with the spindle axis involved in the cutting process. The thread chain function allows multiple thread cutting blocks to be strung together. They are then processed together on the path without motion stop.

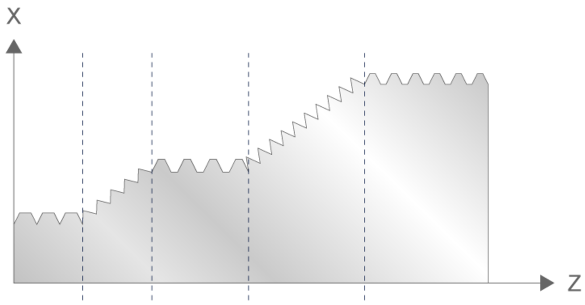


Fig. 10: Thread chain

Release Note

This function is available as of V3.1.3081.11 or V3.1.3118.0

The function is activated either by P-CHAN-00837 [▸ 53] or by the NC command #TURN [▸ 36] [THREAD_CHAIN=1].

Attention

Any override changes in a thread chain are ineffective.

Programing Example

Thread chain with 5 thread cutting blocks

```
%Thread_chain
N010 G74 S1
N020 M03 S2000
N030 G90 G51
N040 #TURN [THREAD_CHAIN = 1]
N050 G0 X100 Z0 Z500
N060 G33 Z100  K5
N070 G33 Z200 X140
N080 G33 Z300
N090 G33 Z350 X100
N100 G33 Z450  K7.5
N110 G0 Z500
N120 M05
N130 M30
```

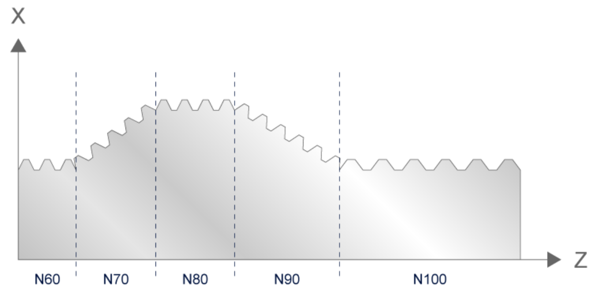


Fig. 11: Thread chain with 5 thread cutting blocks

2.1.8 Constant cutting speed (G96/G97/G196)

Syntax:

G96	Selecting constant cutting speed	modal
G97	Deselecting constant cutting speed, selecting spindle speed	modal, initial state
G196	Maximum spindle speed for G96	G196 non modal max. speed modal

Using the G functions G96, G97 and G196, it is possible to optionally change the interpretation of the S word:

G96	S in [m/min or ft/min *] (cutting speed)
G97	S in [rpm] (spindle speed)
G196	S in [rpm] (max. spindle speed during G96)

* [as of Build V2.11.2032.08 with G70 and P-CHAN-00360 = 1]

When selected with G96, the starting rpm of the spindle is calculated from the programmed cutting speed and the distance of the tool tip to the turning centre point. This distance results from the last (not in the current NC block) programmed position and the reference point offset of the face turning axis. Exactly one face turning axis must be present in the current machining plane (G17, G18, G19).

A cutting speed programmed for G96 using the S word is only valid until it is deselected by G97. With G96, constant cutting speed is only activated when the S word is programmed.

Specifying a maximum spindle speed with G196 in conjunction with the S word is optional and only active during G96. Spindle speed limiting must be programmed before G96 is selected.

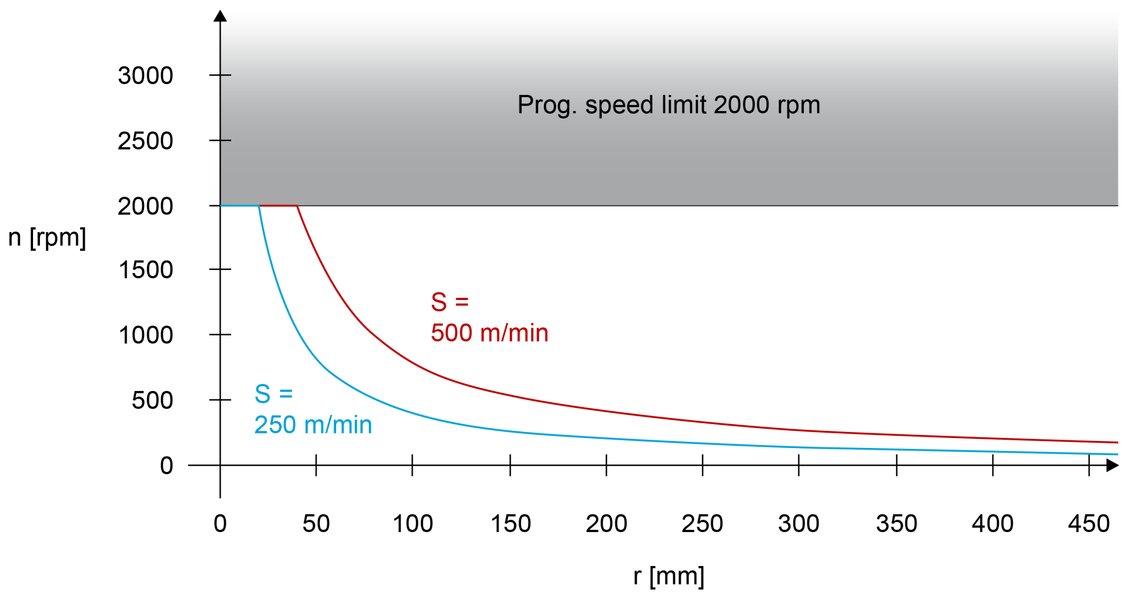


Fig. 12: Spindle speed with active G96

Release Note

Extended G function G196

As of Build V3.1.3057.04

Alternatively, the maximum spindle speed can be programmed as an additional value in [rpm] in conjunction with G196. It is modal.

This syntax permits the programming of G196 and G96 in the same NC block. A separate specific NC block is not required.

Syntax:

G196 = *<Max_spindle_speed>* G196 non modal, max. speed modal

Close to the turning centre point, the programmed maximum spindle speed (G196) or the maximum spindle speed specified in the assigned axis parameters P-AXIS-00212 defines the limits of the constant cutting speed.

When deselected with G97, the last spindle speed set is retained.

Motion blocks of the face turning axis in rapid traverse (G00) lead to an interruption of G96 to prevent undesired speed value changes when the tool is positioned. The next motion block with G01, G02 or G03 cancels suppression of G96.

Programing Example

Constant cutting speed (G96/G97/G196)

```
; X is the face turning axis

N10 M03 S1000 G01 F1500 X100
N20 G196 S6000    ;max. speed 6000 rpm
N30 G96 S63 ;select const. cutting speed 63 m/min,
              ;workpiece radius 100mm corresp. to X coordinates
N40 X80
N50 S4 X50    ;new cutting speed 4m/min; workpiece radius 80mm,
              ;at block end 50mm
N60 G97    ;max. speed 6000 rpm not effective here!
N80 G92 X-10  ;reference point offset in X by -10mm
N90 G96 X60   ;cutting speed from N50 not valid: const.
              ;cutting speed not active, speed 8000 rpm
N100 S25 X70    ;cutting speed 25m/min, workpiece radius 50mm,
              ;(=60mm+BPV), const. cutting speed active
N110 G00 X450 ;rapid traverse: speed remains constant
N115 X70
N120 G01 X40  ;suppress G96 cancelled
N110 M30
```

2.1.9 Constant cutting speed at feedrate per revolution (G96, G95)

When combined with the functions G96 and G95, a constant chip thickness and constant cutting speed (*) are obtained. The path feed is adapted to the spindle speed. When face turning, the following path speed curve results depending on the X position (turning radius). The path feed increases towards the turning centre point.

(*) Provided the dynamics of the spindle and path do not result in limiting effects.

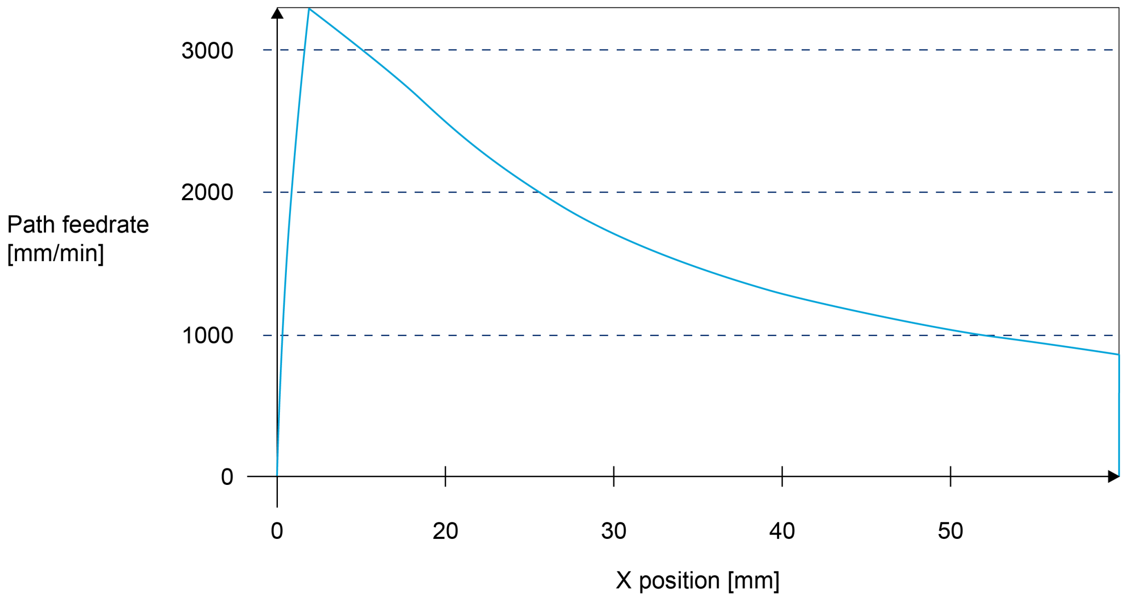


Fig. 13: Path feed for G96 at G95

Programing Example

Constant cutting speed at feedrate per revolution

```
; X... Face turning axis

N10 M03 S2000 G01 X60 Z100 F1500
N20 G196 S2500          ;max. Speed 2500 1/min
N25 G95 F1.5          ;feedrate per revolution 1.5mm/rev
N30 G96 S100            ;const. cutting speed 100m/min
N40 Z49
N50 X10
N60 G00 Z50
N70     X60             ;rapid traverse: speed remains constant
N80     Z48
N90 G01 X10             ;suppress G96 cancelled
N100 G97                ;Deselect const. cutting speed
N110 G94                ;deselect feedrate per revolution
N120 M30
```

2.1.10 Tapping (G331/ G332)

Release Note

This function is available as of CNC Build V3.1.3067.01.

```
Syntax example of tapping in Z direction:
```

G331 Z.. **K**.. *<spindle_name>*..	Thread tapping	modal
G332 Z.. [**K**..] [*<spindle_name>*..]	Thread tapping, retract	modal

G331	Thread tapping
Z..	Thread depth (target point) in the tapping axis in [mm, inch]
K..	Thread pitch in assigned interpolation parameter in [mm/rev, inch/rev]
<spindle_name>..	Spindle speed consisting of spindle name according to P-CHAN-00053 and speed value in [rpm]
G332	Retract from threaded bore (Retract). G332 causes an automatic direction reversal of the spindle on retraction.
Z..	Retract position of tapping axis after tapping in [mm, inch]
K..	Thread pitch in assigned interpolation parameter in [mm/rev, inch/rev]. The thread pitch must be the same pitch as used for the threaded bore assigned in G331. The parameter is optional. If not programmed, the pitch in block G331 applies.
<spindle_name>..	Spindle speed consisting of spindle name according to P-CHAN-00053 and speed value in [rpm]. The parameter is optional. If not programmed, the speed in block G331 applies.

This type of tapping (G331/ G332) requires a position-controlled spindle which is tracked by the CNC synchronous to the path motion. In this case the spindle and the feed motion of the participating axes are matched precisely and dynamically. A compensatory chuck is not required.

The thread type is defined by specifying a sign for thread pitch.

- Pitch without or with positive sign (+): Right-hand thread, e.g. K2 or K+2
- Pitch with negative sign (-): Left-hand thread, e.g. K-2

The thread tapping axis feedrate is a product of the programmed pitch and the spindle speed. The permissible speed limits apply to the internal calculation. An error message is output if these limits are violated.

The feed rate continues to apply after tapping is completed. With the following G331/G332, the feedrate is again calculated from the related programmed or saved values of pitch and spindle speed.

G331/G332 is deselected by selecting a different modal block type (e.g. linear motion G01) and the spindles are released from the coordinated motion. A non-modal block type (e.g. dwell time with G04) does not deactivate G331/ G332.

An error message is output if the pitch or spindle speed with G331/G332 are equal to zero or the tapping axis and pitch parameters fail to match. Valid combinations are X with I, Y with J and Z with K.

M03, M04, M05, M19 cannot be programmed in combination with G331/G332.

Attention

The spindle (or the thread tapping drill) must be at standstill when G331 is selected. This can be achieved by previously programming M05 (Stop spindle) or M19 with S.POS (Position spindle).

Programing Example

Tapping (G331/ G332)

Tap right-hand thread with pitch 2 mm, thread depth 50 mm, spindle speed S 200 rpm, Z is tapping axis:

```
;…
G01 F2000 G90 X0 Y0 Z0 ; position axes
M19 S.POS=0 M3 S100    ; stop and position spindle
;…
G331 Z-50 K2 S200     ; tap in Z
G332 Z10 K2 S200 ; retract
G01 F1000 X50         ; reposition, deselect tapping
G331 Z-50 K2 S200     ; tap in Z
G332 Z10  K2 S400      ; retract at increased speed
G01 F1000 X100         ; reposition, deselect tapping
G331 Z-50 K2 S200     ; tap in Z
G332 Z10               ; retract, K and S from G331
G01 F1000 X150         ; reposition, deselect tapping
;…
```

Tap right-hand thread with pitch 1.5 mm, thread depth 60 mm, spindle speed S 150 rpm, X is tapping axis:

```
;…
G01 F2000 G90 X100 Y0 Z0 ; position axes
M19 S.POS=0 M3 S100    ; stop and position spindle
;…
G331 X40 I1.5 S150     ; tap in X
G332 X110 I1.5 S150 ; retract
:
```

Programing Example

Tap at relative speed

```
%Tap at relative speed

N010 G91 G19 G0 X100 M03 S2000
N020 S2[MC_GearIn Master=S1 \ ; couple tool
RN=1 RD=1 Mode=256 \          ; spindle S2 to the
PhaseShift=1800000 WAIT_SYN] ; main spindle S1

N030 #MAIN SPINDLE[S2]         ; main spindle tool spindle S2
N040 G331 Z-100 K1.5 S200      ; tap right-hand thread
N050 G332 Z100 K1.5 S200      ; retract from threaded bore
N060 G01 X300 F1000
N070 S2[MC_GearOut WAIT_SYN]   ; release coupling to main spindle
N080 #MAIN SPINDLE[S1]
N090 M30
```

2.1.11 Thread drilling without compensating chuck (G63)

```
Syntax example of tapping in Z direction:
```

G63 **Z**.. **F**.. *<spindle_name>*.. modal

G63	Thread tapping
Z..	Thread depth (target point) in the tapping axis in [mm, inch]
F..	Feed rate in [mm/min, m/min, inch/min]
<spindle_name>..	Spindle speed consisting of spindle name according to P-CHAN-00053 and speed value in [rpm]

With this kind of tapping (G63) the position-controlled spindle is tracked by the CNC synchronously to the path motion. In this case the spindle and the feed motion of the participating axes are matched precisely and dynamically. A compensatory chuck is not required. The programmed feed rate must match the programmed spindle speed and the thread pitch and is calculated as follows:

Feed rate F [mm/min] = speed S [rpm] * pitch [mm/rev]

G63 is deselected by the selecting a different modal block type (e.g. linear motion G01). A non-modal block type (e.g. dwell time with G04) does not deactivate G63.

The path feed rate (F word) and spindle speed (S word) do not necessarily need to be specified in the same NC block as G63. The feed rate calculation must always be based on the last values programmed.

An error message is output if the path feed rate or spindle speed are equal to zero with G63 is selected.

M03, M04, M05, M19 cannot be programmed in combination with G331/G332.

Attention

The spindle (or the thread tapping drill) must be at standstill when G63 is selected. This can be achieved by previously programming M05 (Stop spindle) or M19 with S.POS (Position spindle).

Cutting a left-hand thread or movement out of a thread hole is programmed with a **negative S value**.

In C axis mode, the gear stage can be defined using the parameter P-AXIS-00052.

Programing Example

Tapping (G63)

Tap a right-hand thread with pitch 1.25 mm, thread depth 50 mm. At a programmed spindle speed S of 200 rpm the calculated feedrate is:

F = 200*1.25 = 250 mm/min

```
;…
G01 F2000 G90 X0 Y0 Z0 ; position axes
M19 S.POS=0 M3 S100    ; stop and position spindle
;…
G63 Z-50 F250 S200     ; tap
    Z0 S-200           ; retract from threaded bore
G01 F1000 X100         ; reposition, deselect tapping
:
```

Programing Example

Tapping (G63)

```
%Tapping_G63

N05 X0 Y0 Z0
N10 G91 Z100
N20 M19 S.POS180 M3 S100    ; position spindle

N30 G63 Z-50 F300 S200      ; tap
N40 Z100 S-200              ; retract from threaded bore

N50 G01 X200 F3000     ; reposition, deselect tapping

N60 G63 Z-70 F300 S200      ; tap
N70 Z100 S-200              ; retract from threaded bore

N80 M05 G01 X300 F1000
N90 M30
```

A thread can also be executed in a rotating workpiece using G63 or G331/G332. The thread tapping drill feedrate results from the speed difference between the tool spindle and the driven thread tapping drill. Before starting the actual tapping, the spindle with the thread tapping drill must be coupled to the workpiece spindle at a synchronous speed. No spindle stop is required to start tapping or to establish the speed coupling. The process can be started on the fly both for the tool and the workpiece spindles.

2.1.12 Thread tapping with spindle actual positions

When tapping a thread, the spindle speed may tend to deviate under load. In order to execute thread tapping in such cases, the movement of the linear axes can be directly coupled to the spindle's actual positions.

Release Note

This function is available as of Build V3.01.3080.04.

The function is activated either with P-CHAN-00761 [▸ 52] or with the NC command #TURN [▸ 36][TAPPING_ACT_POS=1 …]

Example

Thread tapping with spindle actual positions

Set the following parameters:

```
tapping.use_actual_position      1    ( P-CHAN-00761 )
tapping.n_cycles                 5    ( P-CHAN-00762 )
```

NC program

```
%tapping.nc
; tapping command speed S=200rpm,
; right-hand thread with 1.5mm pitch
N05 G0 Z10 X0
; thread tapping with unfiltered actual positions
N10 M19 S.POS=0 M3 S1000
N20 #TURN[TAPPING_ACT_POS=1 TAPPING_N_CYCLES=0]
N30 G63 Z-150 F300 S200
N40 G63 Z10 S-200
N50 G0 X20
; tapping with filtered actual positions
N60 M19 S.POS=0 M3 S1000
N70 #TURN[TAPPING_N_CYCLES=5]
N80 G331 Z-150 K1.5 S200
N90 G332 Z10 K1.5 S200
N100 G0 X40
; tapping with command positions
N110 M19 S.POS=0 M3 S1000
N120 #TURN[TAPPING_ACT_POS=0]
N130 G63 Z-150 F300 S200
N140 G63 Z10 S-200
N160 M30
```

2.1.13 Settings for turning functions (# TURN)

Release Note

This function is available as of CNC Build V3.1.3079.03.

The #TURN command can influence rotary functions.

```
Syntax:
```

#TURN [[**ROT_FEED_CPL=**..] [**TAPPING_ACT_POS=**..] [**TAPPING_N_CYCLES=**..]
[**THREAD_CUT_ACT_SPEED =**..] [**THREAD_CUT_N_CYCLES =**..]
[**THREAD_CUT_IGNORE_OV =**..] [**THREAD_CHAIN =**..]**]**

ROT_FEED_CPL=..	Influence of axis couplings on the revolution feed rate with G95 0 : Axis couplings are not considered (default) 1 : Axis couplings are considered
TAPPING_ACT_POS=..	Enable tapping at actual spindle positions. (analogous to Channel parameters [▸ 52]) 0: The spindle is coupled to the command positions of the linear axis. 1: The linear axes are coupled to the actual spindle positions. Available as of V3.1.3080.04
TAPPING_N_CYCLES=..	Number of filter cycles to filter actual spindle positions. (Analogous to Channel parameters [▸ 52]) Value range: 0 … 20 0: Filter is deactivated. Available as of V3.1.3080.04
THREAD_CUT_ACT_SPEED=..	Enable thread cutting at actual spindle speed. (Analogous to Channel parameters [▸ 52]) 0: The linear axes are coupled to the command spindle speed. 1: The linear axes are coupled to the actual spindle speed. Available as of V3.1.3080.16
THREAD_CUT_N_CYCLES=..	Number of filter cycles to filter actual spindle speed. (Analogous to Channel parameters [▸ 53]) Value range: 0 … 20 0: Filter is deactivated. Available as of V3.1.3080.16
THREAD_CUT_IGNORE_OV=..	During thread cutting G33 [▸ 18] it can be specified whether an evaluation of the speed override should take place (analogous to P-CHAN-00836 [▸ 53]) 0: Override changes are included in the thread cutting block G33 1: Override changes are not included in the thread cutting block G33 Available as of Build V3.1.3081.11 or V3.1.3118.0
THREAD_CHAIN=..	Thread chain: Linking and contouring G33 [▸ 18] thread cutting blocks (analogous to P-CHAN-00837 [▸ 53]) 0: Each thread cutting block G33 is processed separately and synchronised with the spindle axis before every thread cutting operation 1: The following thread cutting blocks are regarded as a thread chain. Contouring takes place between the thread cutting blocks G33 without motion stop. Available as of Build V3.1.3081.11 or V3.1.3118.0

2.2 Functions for lathes / milling machines

These functions can be used for turning and milling work; the participating axes and the spindle axis are interpolated in the coordinated motion.

2.2.1 C axis machining

Axis configuration in the NC channel		
Axis identifier	X, Y, Z, S	
Axis index	0, 1, 2, 3	
Kinematic structure (ID13, ID14, ID 15)		
	Tool axes	**Workpiece axes**
NC axes	X, Y, Z	(S) C

This functionality supplements the existing turning functions and permits the face and lateral surface machining of cylindrical workpieces on lathes and milling machines with revolving base. The workpiece is moved by the rotary axis or spindle (C axis) and the driven tool (e.g. a milling cutter) by the two translatory axes X (or Y) and Z. Specific ettings are required in the parameters P-CHAN-00008, P-AXIS-00015 and P-AXIS-00018 for C axis machining.

Example of a configuration of a C axis in the axis parameters

```
kenngr.achs_typ     0x0002  #Path axis
or
kenngr.achs_typ     0x0004  #Spindle axis
kenngr.achs_mode    0x0204  #MODULO and CAX
```

Facing and lateral surface machining can be described in Cartesian coordinates.

All interpolation types (such as linear, circular or spline interpolation) are supported on the end face and lateral surface. The functionality also permits the machining of path contours running through the turning centre point. The C axis is automatically aligned on lathes.

The 2.5D tool radius compensation can be used with the familiar G commands.

The use of extended dynamic monitoring can specifically prevent dynamic axis characteristics from being exceeded with the C axis function and also with contours running close to the turning centre point.

The main axes for all machining modes are X, Y (depending on machine type), Z and C.

2.2.1.1 Exchange spindles in coordinated motion (# CAX, #CAX OFF)

This "basic mode" is required in particular for C axis machining on lathes because in this case the position-controlled spindle has to be converted into a rotary path axis (e.g "C").

Notice

C axis machining can also be executed on milling machines or machining centres which are designed with rotary workpiece fixtures (e.g. turntable). In this case, it is not necessary to select #CAX.

The three physical axes X, Y, Z and the C axis replaced in coordinated motion can be directly programmed. Linear axes are programmed in Cartesian coordinates and the C axis in angle units.

Radius and diameter programming depends on G52/G51.

Two linear axes define the main plane: ZX (G18) or YZ (G19).

`Syntax:`

#CAX [**[** [<*main_spindle_name*>,] <*C_axis_name*> **]**]

<*main_spindle_name*>	Only the main spindle name can be programmed according to P-CHAN-00053. If a spindle other than the C axis is used, it must first be declared as the main spindle (see programming example in Section [PROG// Changing the main spindle]). Otherwise, an error message is output.
<*C_axis_name*>	Freely definable name of the C axis in the NC program. If no C axis name is programmed, the default name from P-CHAN-00010 is used.

The main plane (circular interpolation, tool radius compensation, etc.) remains the same as before activation of the C axis.

An error is generated if a command for this spindle (M3, M4, M5, etc.) is programmed although the axis is still declared as a C axis in the coordinated motion.

The C axis is deselected, i.e. the axis is released to the spindle interpolator, by the following:

`Syntax:`

#CAX OFF

Programing Example

C axis machining

Exchange spindles in coordinated motion

```
;…
#CAX                        ; Assuming: default C axis is "C"
G01 G90 X50 Z10 C90 F200
#CAX OFF                    ; deselect C axis mode
;...
#CAX[S, C] or #CAX[C]     ; Assuming: main spindle is "S"
G01 G90 X50 Z10 C90 F200
#CAX OFF
;...
#MAIN SPINDLE [S2]          ; "S2" becomes new main spindle "S"
#CAX[S, C]                  ; select C axis mode
G01 G90 X50 Z10 C90 F200
;...
#CAX OFF                    ; deselect C axis mode
;...
#CAX[S3, C]; Error, "S3" is not the main spindle
```

2.2.1.2 Face machining (#FACE, #FACE OFF)

This mode is selected for lathes and machining centres. The desired contour on the face is programmed in millimetres or inches using a virtual Cartesian coordinate system.

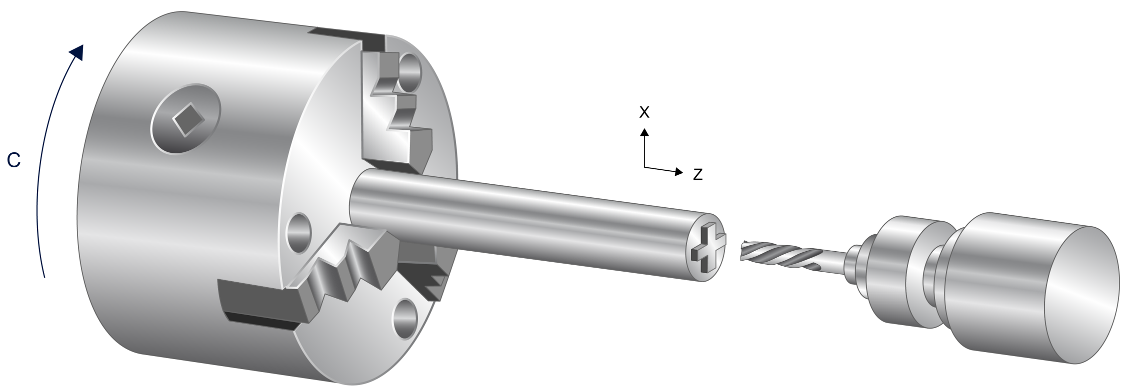


Fig. 14: Face machining

Notice

As of CNC build V3.00, the parameter P-CHAN-00262 [▸ 59] must be assigned with the kinematic ID used, depending on P-CHAN-00008 [▸ 59], in order to perform face machining applications.

a) For face transformation 1 with P-CHAN-00008=1 - **ID 13**

b) For face transformation 2 with P-CHAN-00008=2 - **ID 14**

The three logical axes X, Y (or C) and Z are provided to program the contour on the face in Cartesian coordinates.

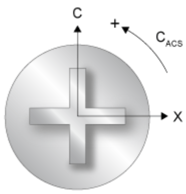


Fig. 15: Front view of face machining process

The figure below shows each of the main planes in face machining. Only the G17 plane is of technological importance.

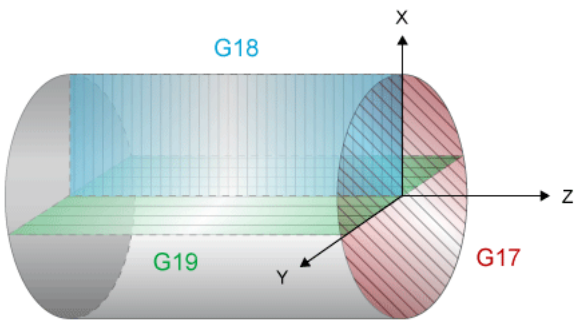


Fig. 16: Main planes of face machining

For more setting options for face machining, see [KITRA// KIN_TYP_13/14]

`Syntax:`

#FACE [*<name of 1st main axis>*, *<name of 2nd main axis>* **]**

<name of 1st main axis>	Name of the first main axis according to the current main plane.
<name of 2nd main axis>	Name of the second main axis according to the current main plane (virtual Cartesian axis).

When selected. the main plane (circular interpolation, tool radius compensation, etc.) is always defined by the 1st and 2nd main axes (G17). It is not permitted to change the main plane with G18, G19 while face machining is active.

Notice

Programmed tracking axes are not affected by the transformation.

This mode is deselected by:

`Syntax:`

#FACE OFF

Use #FACE OFF to revert to the previously active state. This means that the last active main plane is selected automatically and the last active axis offsets are restored.

Programing Example

Programming example for lathes

Example with axis name "C" for 2nd main axis

```
;…
#CAX[S, C]                  ;Assuming main spindle is "S"
#FACE[X, C]                 ;Select face machining
;…
G01 X40 C-30 Z50 F1000        ;Pre-position
G01 Z30                     ;Feed
G01 X10 C40                 ;Travel contour
G01 Z50                     ;Retract
;…
#FACE OFF
#CAX OFF
;…
M30
```

Example with axis name "Y" for 2nd main axis.

Note: No other axis with the identical name "Y" may exist in NC channel.

```
;…
#CAX[S, Y]                  ;Assuming main spindle is "S"
#FACE[X, Y]                 ;Select face machining
;…
G01 X40 Y-30 Z50 F1000        ;Pre-position
G01 Z30                     ;Feed
G01 X10 Y40                 ;Travel contour
G01 Z50                     ;Retract
;…
#FACE OFF
#CAX OFF
;…
M30
```

Programing Example

Programming example for machining centres

The rotary axis (workpiece axis) in the channel is "C2". It is not necessary to program the #CAX command.

```
;…
#FACE[X, C2]                 ;Select face machining
;…
G01 X40 C2=-30 Z50 F1000     ;Pre-position
G01 Z30                      ;Feed
G01 X10 C2=40                ;Travel contour
G01 Z50                      ;Retract
;…
#FACE OFF
;…
M30
```

2.2.1.2.1 Face machining with 2 rotary axes (#FACE 2ROT, #FACE OFF)

Kinematic ID 203 is required for face machining with two rotary axes, The tool is then supported by the rotary axis Y and the translatory axis Z. The workpiece is aligned by the rotary axis C.

Transformation can be switched by the command #FACE 2ROT. The user programs X, Y, Z on the tool face in the Cartesian system.

The C axis must have a modulo range of 0° to 360°, see P-AXIS-00126 and P-AXIS-00127.

Notice

In order to use this face machining, the value 203 must be assigned to P-CHAN-00262 [▸ 59] for this transformation.

Syntax:

#FACE 2ROT [AX1=<*name*> **AX2=**<*name*> **AX3=**<*name*>**] | [AXNR1=.. AXNR2=.. AXNR3=..]**

AX1=<*Name*>	Designation of first axis
AX2=<*Name*>	Designation of second axis
AX3=<*Name*>	Designation of third axis
AXNR1=..	Logical axis number P-AXIS-00016 [▸ 66] of first axis
AXNR2=..	Logical axis number P-AXIS-00016 [▸ 66] of second axis
AXNR3=..	Logical axis number P-AXIS-00016 [▸ 66] of third axis

This mode is deselected by:

Syntax:

#FACE OFF

2.2.1.3 Surface machining (#CYL, #CYL OFF)

This mode can be selected for lathes and machining centres. The desired contour on the cylindrical surface is programmed in millimetres or inches using a virtual coordinate system.

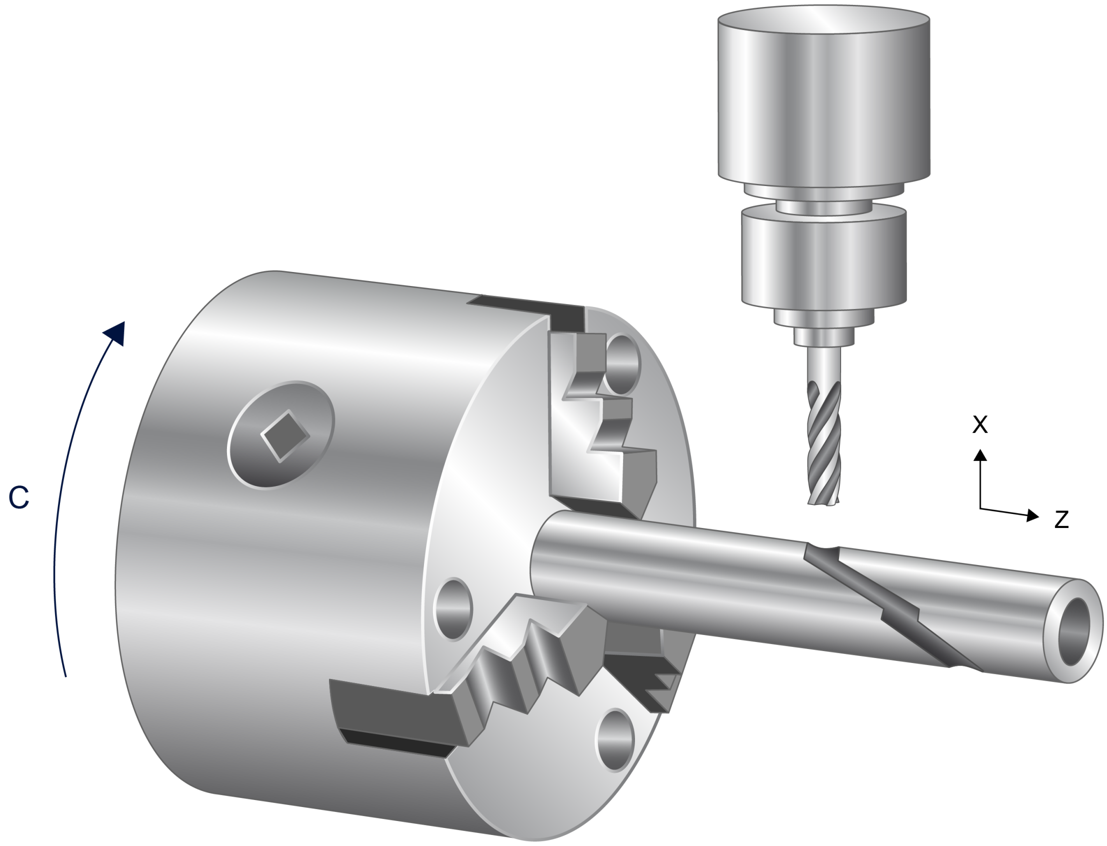
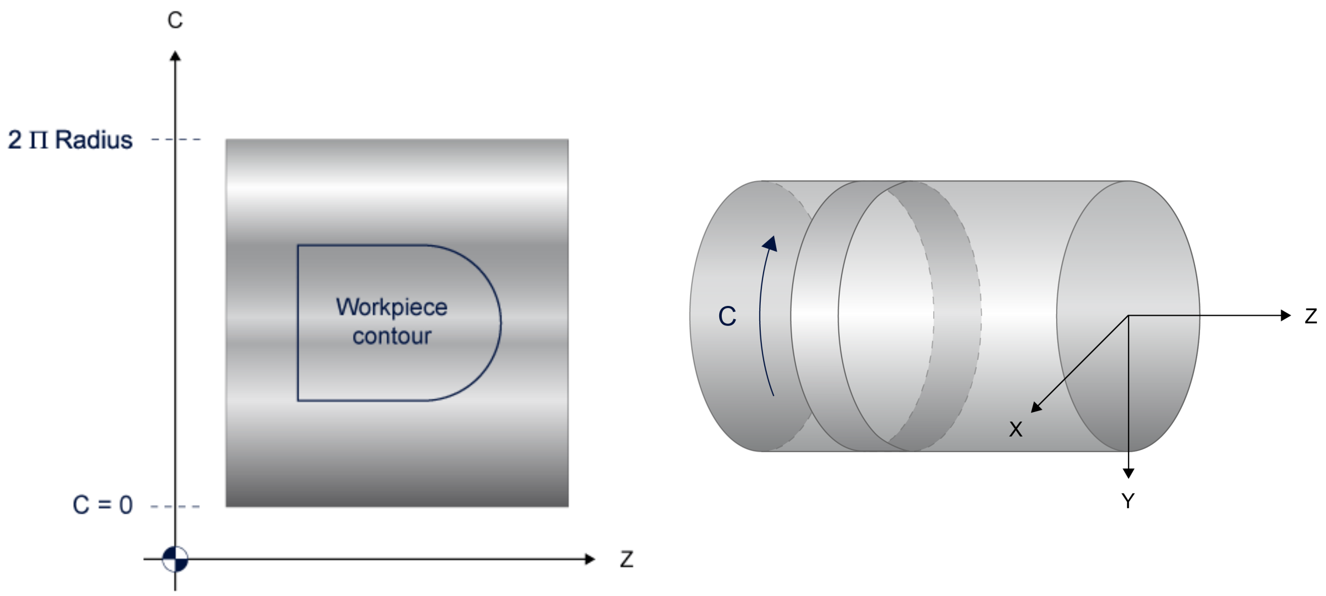


Fig. 17: Lateral surface machining

Notice

As of CNC build V3.00, the parameter P-CHAN-00262 [▸ 59] must be assigned with the vale 15 for this transformation in order to perform lateral surface machining applications.

Depending on the machine type, the three logical axes X, Y, Z are provided to program the contour in Cartesian coordinates on the lateral surface. As a general rule, the Y axis is not present on lathe-only machines. The workpiece radius must also be included in the programming as reference radius R.



The main plane in lateral surface machining is formed by Z-C.

Lateral surface machining in G17

An axis configuration Z-C is formed by specifying the first and second main axes with #CYL [...]. This implicitly defines a main plane in G17. The reference radius must also be specified.

Syntax:

#CYL [<1st_main_axis_name>**,** <2nd main_axis_name>**,** <3rd_main_axis_name>..]

<1st_main_axis_name>	Name of the first main axis according to the current main plane.
<2nd main_axis_name>	Name of the second main axis according to the current main plane (virtual linear axis, development).
<3rd_main_axis_name>.	Axis name of the third main axis according to the current main plane with specification of the reference radius in [mm, inch].

Notice

Programmed tracking axes are not affected by the transformation. It is not permitted to change the main plane with G18, G19 while lateral surface machining is active

This mode is deselected by:

Syntax:

#CYL OFF

Use #CYL OFF to revert to the previously active state. This means that the last active main plane is selected automatically and the last active axis offsets are restored.

Programing Example

Programming example for lathes, programming in G17 with Z-C

Example with axis name "C" for 2nd main axis

```
…
#CAX [S, C]        ;Assuming “S” is main spindle
G01 X60 C45 F800   ;Feed and positioning movement; X:60mm C:45°
#CYL [Z, C, X60]   ;Select lateral surface machining
G00 G90 Z0 C0      ;Z: 0mm C:0mm!
G01 C100 F500
G02 Z100 R50
G01 C0
Z0
…
#CYL OFF
#CAX OFF
M30
```

Lateral surface machining in G19

Use #CYL LATERAL [...] to create an axis configuration which, in combination with G19, allows programming in the virtual coordinate system C-Z. The reference radius must also be specified.

Syntax:

#CYL LATERAL [RADIUS=..]

RADIUS=..	Specify reference radius in [mm, inch].

Notice

Programmed tracking axes are not affected by the transformation. After #CYL LATERAL, the use of G17 or G18 is also permitted; this may be necessary for special machining cases.

This mode is deselected by:

Syntax:

#CYL OFF

Use #CYL OFF to deselect lateral surface machining and restore the previously active axis configuration with the associated axis offsets. The currently valid main plane remains active.

Programing Example

Programming examples for lathes, programming in G19 with C-Z

```
%cyl_lat_A
N020 G00 X0 Y0 Z0
N020 #CAX [S,C]
N030 G00 X0 Y0 Z100 C0
N040 #CYL LATERAL [RADIUS=35]  ;Deselect lateral surface machining
N060 G19                       ;Select G19 plane
N070 G01 G90 Z0 C0 F5000
N080 G01 G90 X10   F5000

N090 $FOR P2=1, 5, 1
N100 P3=P2*4
N110 P4=P3+2
N120 G01 G91 C-P3
N130 ZP3
N140 C[2*P3]
N150 Z-P3
N160 G90 C0
N170 G91 ZP4
N180 $ENDFOR

N190 $FOR P2=1, 5, 1
N200 P3=P2*4
N210 P4=P3*2+2
N220 G90 G02 KP3
N230 G91 G01 ZP4
N240 $ENDFOR

N270 #CYL OFF
N280 #CAX OFF
N290 M30
```

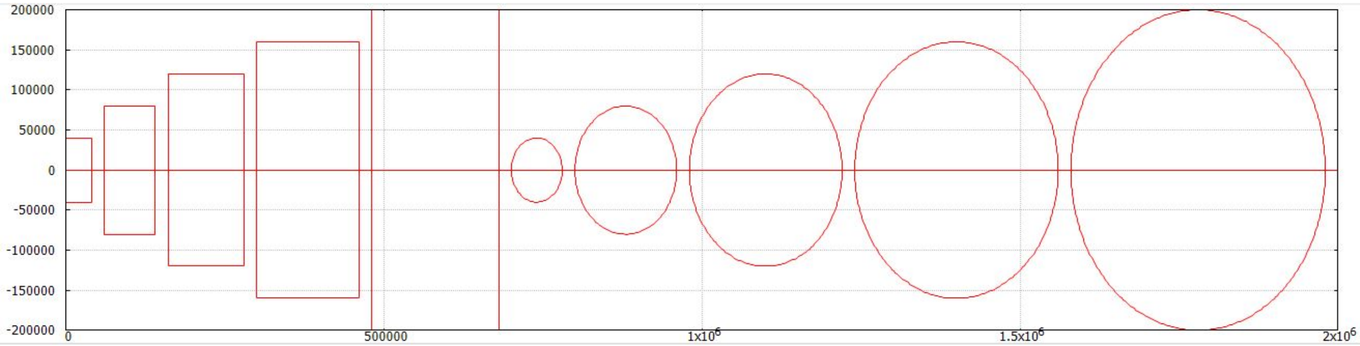


Fig. 18: Example 1 for #CAX LATERAL with G19

```
%cyl_lat_B
N020 G00 F2000 X0 Y0 Z0
N030 #CAX [S, C]
N040 #CYL LATERAL [RADIUS=20]
N060 G19
N070 G01 F1000 Z0 C0 X0  G161
N080 G02 J30  K0   C60 F2000
N090 G02 J30  K0   C0

N100 G02 J0   K30  Z60
N110 G02 J0   K30  Z0

N120 G03 J-30 K0   C-60
N130 G03 J-30 K0   C0

N140 G03 J0   K-30 Z-60
N150 G03 J0   K-30 Z0

N180 #CYL OFF
N190 #CAX OFF
N210 M30
```

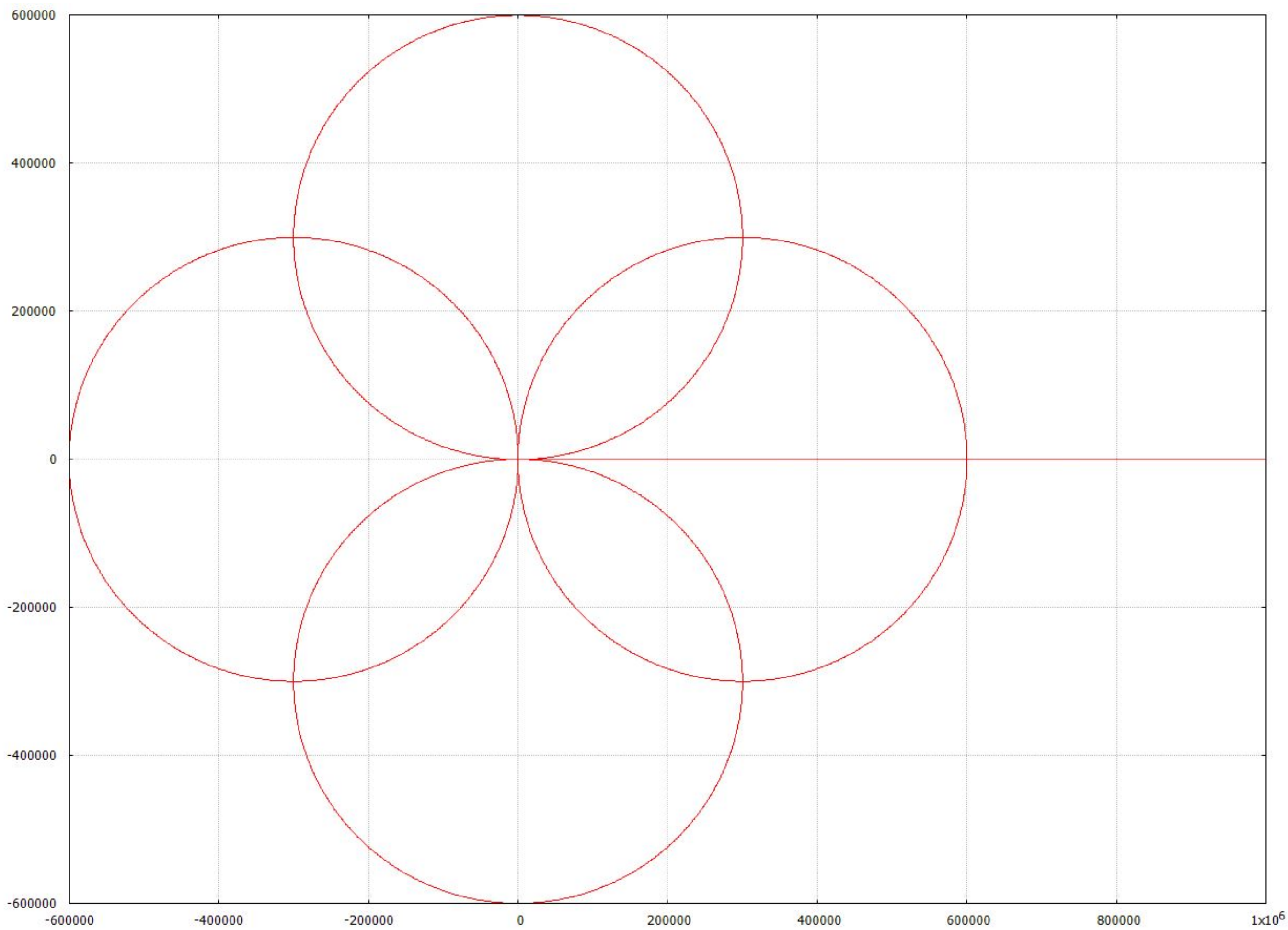


Fig. 19: Example 2 for #CYL LATERAL with G19

2.2.1.3.1 Lateral surface machining with 2 rotary axes (#CYL 2ROT, #CYL OFF)

Turning work in the Cartesian system. The tool is supported by the translatory axes X, Z and the rotary axis Y. The workpiece is aligned by the rotary axis C.

Transformation can be switched by the command #CYL 2ROT. The user programs X, Y, Z in the Cartesian system. In addition, the dimension C of the tool angle is programmed in the X-Y plane. When C=0, the tool is parallel to X and vertical on Y.

Notice

In order to use this lateral surface machining, the value 202 must be assigned to P-CHAN-00262 [▸ 59] for this transformation.

The kinematic is available in two variants.

1. In the LEFT variant (HD10 = 0) the C axis is located to the left of the Y axis. The Y axis points to the left in zero position. When transformation is selected, the X axis must be located to the right of the C axis.
2. In the RIGHT variant (HD10 = 1) the C axis is located to the right of the Y axis. The Y axis points to the right in zero position. When transformation is selected, the X axis must be located to the left of the C axis.

The C axis must have a modulo range of 0° to 360°, see P-AXIS-00126 and P-AXIS-00127.

`Syntax:`

#CYL 2ROT [AX1=<*name*> **AX2=**<*name*> **AX3=**<*name*>**] | [AXNR1=.. AXNR2=.. AXNR3=..]**

AX1=<*Name*>	Designation of first axis
AX2=<*Name*>	Designation of second axis
AX3=<*Name*>	Designation of third axis
AXNR1=..	Logical axis number Axis parameter [▸ 66] of first axis
AXNR2=..	Logical axis number Axis parameter [▸ 66] of second axis
AXNR3=..	Logical axis number Axis parameter [▸ 66] of third axis

This mode is deselected by:

`Syntax:`

#CYL OFF

3 Parameter

3.1 Overview

Constant	Description
SLOPE_VB_MAX	Maximum axis speed that can be parameterised
ACHS_NAME_LAENGE	Maximum length of the axis designation

ID	Parameter	Description
P-CHAN-00082	spdl_anzahl	Number of spindles in the channel
P-CHAN-00036	log_achs_nr	Spindle axis number
P-CHAN-00007	bezeichnung	Spindle designation
P-CHAN-00010	default_ax_name_of_spindle	Name of the spindle as channel axis
P-CHAN-00043	m19_synch	Synchronisation type of the M19 function
P-CHAN-00045	m3_synch	Synchronisation type of the M03 function
P-CHAN-00047	m4_synch	Synchronisation type of the M04 function
P-CHAN-00049	m5_synch	Synchronisation type of the M05 function
P-CHAN-00081	s_synch	Synchronisation method of the S function
P-CHAN-00051	main_spindle_ax_nr	Definition of main spindle
P-CHAN-00052	main_spindle_gear_change	Enabling gear changes
P-CHAN-00053	main_spindle_name	Main spindle name
P-CHAN-00074	range_way	Search direction in the speed range table
P-CHAN-00004	autom_range	Automatic gear speed change
P-CHAN-00058	range_table[j].min_speed	Lower limit of speed ranges
P-CHAN-00055	range_table[j].max_speed	Upper limit of speed ranges
P-CHAN-00008	cax_face_id	Face machining variant (turning/milling)
P-CHAN-00094	kinematik[i].param[j]	Kinematic offsets for face/lateral surface machining up to Build V3.00
P-CHAN-00262	trafo[j].id	Kinematics ID of lateral surface machining
P-CHAN-00263	trafo[j].param[k]	Kinematic offsets for face/lateral surface machining as of Build V3.00

ID	Parameter	Description
P-CHAN-00761	tapping.use_actual_position	Tapping with actual positions of the spindle
P-CHAN-00762	tapping.n_cycles	Number of filter cycles to filter actual spindle positions
P-CHAN-00834	thread_cutting.use_actual_speed	Thread cutting with actual spindle speed
P-CHAN-00835	thread_cutting.n_cycles	Number of filter cycles to filter actual spindle speed
P-CHAN-00836	thread_cutting.ignore_override	Influence of velocity override with thread cutting G33
P-CHAN-00837	thread_cutting.thread_chain	Thread cutting of a thread chain

ID	Parameter	Description
P-AXIS-00015	achs_mode	Axis mode
P-AXIS-00016	achs_nr	Axis number
P-AXIS-00018	achs_typ	Axis type
P-AXIS-00058	durchm_prog_abs	Diameter programming (G51) at G90.
P-AXIS-00059	durchm_prog_rel	Diameter programming (G51) at G91.
P-AXIS-00220	vb_regelgrenze	Limit velocity of the measuring system
P-AXIS-00216	vb_min_null	Limit value for speed 0

3.2 Description

3.2.1 Channel parameters

Overview

This section describes the channel-specific parameters for configuring turning functions and rotary spindles.

The spindle configuration defined in the channel parameter list is the default assignment which is provided after the controller is started up.

P-CHAN-00761	Tapping with actual positions of the spindle
Description	An additional function for tapping can be enabled to take into account the actual spindle speed. The linear axes are then coupled to the actual spindle position. This also permits tapping when the spindle speed drops due to the load.
Parameter	tapping.use_actual_position
Data type	BOOLEAN
Data range	0: The spindle is coupled to the command positions of the linear axes. 1: The linear axes are coupled to the actual spindle positions.
Dimension	---
Default value	0
Remarks	Parameter available as of V3.1.3080.04

P-CHAN-00762	Number of filter cycles to filter actual spindle positions
Description	The actual spindle positions may be noisy under certain circumstances. This parameter activates a filter to smooth the actual positions, The parameter specifies the number of filter smoothing cycles. The value 0 deactivates the filter.
Parameter	tapping.n_cycles
Data type	UNS16
Data range	0 <= n_cycles <= 20
Dimension	---
Default value	0
Remarks	P-CHAN-00762 only acts when tapping with actual spindle positions is active (P-CHAN-00761 [▸ 52]). Parameter available as of V3.1.3080.04

P-CHAN-00834	Thread cutting with actual spindle speed
Description	An additional function for thread cutting can be enabled to include the actual spindle speed. Here, the linear axes are coupled to the actual spindle position. This means that threads can also be tapped when the spindle speed drops due to the load.
Parameter	thread_cutting.use_actual_speed
Data type	BOOLEAN
Data range	0: The linear axes are coupled to the command spindle positions 1: The linear axes are coupled to the actual spindle positions.
Dimension	---
Default value	0
Remarks	Available as of V3.1.3080.17

P-CHAN-00835	**Number of filter cycles to filter actual spindle speed**
Description	The actual spindle speed may be noisy under certain circumstances. To smooth the actual speed, this parameter can activate a filter. The parameter specifies the number of filter cycles that are used for smoothing.. The value 0 deactivates the filter.
Parameter	thread_cutting.n_cycles
Data type	UNS16
Data range	0 <= n_cycles <= 25
Dimension	---
Default value	0
Remarks	P-CHAN-00835 only acts when thread cutting is active at the actual spindle speed. (Channel parameters [52]). Available as of V3.1.3080.17

P-CHAN-00836	**Influence of velocity override with thread cutting G33**
Description	This parameter defines the handling of changes to override with thread cutting blocks G33 [18].
Parameter	thread_cutting.ignore_override
Data type	BOOLEAN
Data range	0: Velocity override is effective with thread cutting blocks G33. 1: Velocity override is not evaluated with thread cutting blocks G33.
Dimension	----
Default value	0
Remarks	Available as of Build V3.1.3081.11 or V3.1.3118.0

P-CHAN-00837	**Thread cutting of a thread chain**
Description	By default, consecutive thread cutting blocks are processes individually. If this parameter is set, consecutive thread cutting blocks are identified as a thread chain. Contouring between thread cutting blocks then takes place without any motion stops.
Parameter	thread_cutting.thread_chain
Data type	BOOLEAN
Data range	0: Consecutive thread cutting blocks G33 [18] are processed individually 1: Consecutive thread cutting blocks G33 [18] are identified as a thread chain
Dimension	----
Default value	0
Remarks	Available as of Build V3.1.3081.11 or V3.1.3118.0

3.2.1.1 Spindle

P-CHAN-00082	**Number of configured spindles in NC channel**
Description	This element specifies the total number of existing position-controlled and speed-controlled spindles. The number of spindles must be identical with the entered spindles (spindel[i].*).
Parameter	spdl_anzahl
Data type	UNS16
Data range	0 ≤ spdl_anzahl ≤ 6 (application-specific)
Dimension	----
Default value	0
Remarks	Parameterisation example: A position-controlled and a speed-controlled spindle are to be configured. *spdl_anzahl 2*

P-CHAN-00010	**Spindle name within a path compound**
Description	If a spindle axis is changed with specific machining modes (e.g. C axis mode) in the path compound, it can be addressed by this default name (e.g. C1).
Parameter	default_ax_name_of_spindle
Data type	STRING
Data range	Maximum 16 characters (length of spindle designation, application-specific)
Dimension	----
Default value	*
Remarks	The designation of spindles in the path compound must start with the letters A, B, C, U, V, W, X, Y, Z or Q. After that, all letters and digits are possible. The axis designation must be unique. It may not be identical with the configured name of a channel axis (P-CHAN-00006). Parameterisation example: The C axis receives the designation C1. *default_ax_name_of_spindle C1* * Note: The default value of variables is a blank string.

P-CHAN-00051	**Logical axis number of the main spindle in the NC channel**
Description	This parameter assigns the definition of the main spindle. The logical axis number of one of the spindles which is configured in the channel parameters list is entered here. After the controller starts up, this spindle becomes the main spindle. However, any other spindle in the system can be declared to be the main spindle using a command in the part program (#MAIN SPINDLE [PROG]).
Parameter	main_spindle_ax_nr
Data type	UNS16
Data range	1 ... MAX(UNS16)
Dimension	----
Default value	0
Remarks	Parameterisation example: The spindle with logical axis number 6 is the main spindle. *main_spindle_ax_nr 6*

P-CHAN-00053	**Designation of the main spindle in the NC channel**
Description	Besides the logical axis number, a name must be assigned to the main spindle by which it can be addressed in the subroutine. This parameter therefore assigns an axis name to the main spindle. The axis name is freely selectable, but the first character must be an 'S'.
Parameter	main_spindle_name
Data type	STRING
Data range	Maximum 16 characters (length of spindle designation, application-specific)
Dimension	----
Default value	*
Remarks	The designation of the main spindle cannot be changed in the subroutine. However, if the main spindle is changed using #MAIN SPINDLE (see [PROG]), the new main spindle is assigned. Parameterisation example: The main spindle (logical axis number 6) is programmed in the subroutine with the name 'S'. `main_spindle_ax_nr      6` **`main_spindle_name       S`** `* Note: The default value of variables is a blank string.`

P-CHAN-00036	Logical axis number of a spindle in NC channel
Description	The logical axis number is entered in this parameter. The logical axis name is unique throughout the entire system. The logical axis number assigns the spindle name in the NC program to the axis data (see axis parameters lists in [AXIS]). Therefore only logical numbers make sense if they are known to the NC program. The logical axis number "0" is not permitted.
Parameter	spindel[i].log_achs_nr
Data type	UNS16
Data range	1... MAX(UNS16)
Dimension	----
Default value	-
Remarks	A logical axis number may not be assigned several times. A logical axis number may not be simultaneously configured as path axis and spindle. If this is the case, the plausibility check of the channel parameters generates an error message at start-up.

P-CHAN-00007	Name of a spindle in the NC channel
Description	This parameter defines the default name to address the spindle in the NC program. Please note that as long as a spindle is the main spindle, it can only be programmed using the main spindle name. The spindle name is a string.
Parameter	spindel[i].bezeichnung
Data type	STRING
Data range	Maximum 16 characters (length of spindle designation, application-specific)
Dimension	----
Default value	*
Remarks	The spindle names must start with the letter ‘S’. After that, all letters and digits are possible. Spindle names must be unique. Parameterisation example: Configuration of a 1-channel system with 3 spindles. After start-up, spindle 'S1' with logical axis number 6 is the main spindle. It is addressed by the spindle name 'S'. The spindles with logical axis numbers 11 and 30 are programmed by their default names 'S2' and 'S3'.

```
spdl_anzahl                    3
:
main_spindle_ax_nr             6-> -> ->-
main_spindle_name              S ->-      /
#                                   /     /
spindel[0].bezeichnung         S1-<-      /
spindel[0].log_achs_nr         6-< -< -<-
:
spindel[1].bezeichnung         S2
spindel[1].log_achs_nr         11
:
spindel[2].bezeichnung         S3
spindel[2].log_achs_nr         30
* Note: The default value of variables is a blank string.
```

P-CHAN-00052	**Enable mechanical gear change of main spindle**
Description	This parameter enables or disables gear changes for the main spindle.
Parameter	main_spindle_gear_change
Data type	BOOLEAN
Data range	0: Spindle gear changes disabled for the main spindle 1: Spindle gear changes enabled for the main spindle
Dimension	----
Default value	0
Remarks	The M functions to select the gear speeds of the main spindle M40–45 are activated by the parameter P-CHAN-00052 in the channel parameter list. The M functions M40–45 can be freely used if gear changes are disabled.

P-CHAN-00074	**Direction of range selection for spindle gear change**
Description	This parameter defines whether the lower or the higher gear range is selected when speed ranges overlap. If 'range_way' = 0 the search starts from the lowest speed range, if 'range_way' > 0 from the highest. The correct speed (gear) range is the one where the programmed spindle speed is first found.
Parameter	spindel[i].range_way
Data type	UNS16
Data range	0 ≤ range_way < MAX(UNS16)
Dimension	----
Default value	0
Remarks	Parameterisation example: The search starts from the lowest speed range. *spindel[0].range_way 0 (from bottom to top)*

P-CHAN-00004	**Automatic range selection for spindle gear change**
Description	If the spindle gear range is to be be automatically determined by the NC kernel, this parameter must be set to 1. In this case the M functions M40 to M45 need not be programmed. This means that the correct gear range is determined implicitly by the programmed speed (S word).
Parameter	spindel[i].autom_range
Data type	BOOLEAN
Data range	0/1
Dimension	----
Default value	0
Remarks	Parameterisation example: The automatic range selection is enabled. *spindel[0].autom_range 1*

P-CHAN-00058	Minimum spindle speed of a speed range (spindle gear change)
Description	The speed ranges of a spindle may be defined with or without overlap. If one range is not used, the corresponding values must be set to zero in the table.
Parameter	spindel[i].range_table[j].min_speed
Data type	UNS16
Data range	0 ≤ min_speed ≤ MAX(UNS16)
Dimension	rpm
Default value	0
Remarks	Parameterisation example: Definition of a speed range table for 6 ranges. Only the first four ranges are used. `spindel[0].range_table[0].min_speed 50` `spindel[0].range_table[0].max_speed 560` `spindel[0].range_table[1].min_speed 400` `spindel[0].range_table[1].max_speed 800` `spindel[0].range_table[2].min_speed 700` `spindel[0].range_table[2].max_speed 3360` `spindel[0].range_table[3].min_speed 3361` `spindel[0].range_table[3].max_speed 4000` `spindel[0].range_table[4].min_speed 0` `spindel[0].range_table[4].max_speed 0` `spindel[0].range_table[5].min_speed 0` `spindel[0].range_table[5].max_speed 0`

P-CHAN-00055	Maximum spindle speed of a speed range (spindle gear change)
Description	The speed ranges of a spindle may be defined with or without overlap. If one range is not used, the corresponding values must be set to zero in the table.
Parameter	spindel[i].range_table[j].max_speed
Data type	UNS16
Data range	0 ≤ max_speed ≤ MAX(UNS16)
Dimension	rpm
Default value	0
Remarks	Parameterisation example: Definition of a speed range table for 6 ranges. Only the first four ranges are used. `spindel[0].range_table[0].min_speed 50` `spindel[0].range_table[0].max_speed 560` `spindel[0].range_table[1].min_speed 400` `spindel[0].range_table[1].max_speed 800` `spindel[0].range_table[2].min_speed 700` `spindel[0].range_table[2].max_speed 3360` `spindel[0].range_table[3].min_speed 3361` `spindel[0].range_table[3].max_speed 4000` `spindel[0].range_table[4].min_speed 0` `spindel[0].range_table[4].max_speed 0` `spindel[0].range_table[5].min_speed 0` `spindel[0].range_table[5].max_speed 0`

3.2.1.2 Kinematics

P-CHAN-00008	**Machine ID with C axis face machining**
Description	This parameter defines the type of machine used for face machining.
Parameter	cax_face_id
Data type	UNS16
Data range	1: Lathe (automatic orientation of the rotary axis in the centre of rotation) 2: Milling machines (no orientation)
Dimension	----
Default value	0
Remarks	Parameterisation example: Face machining takes place on a milling machine. *cax_face_id 2*

As of CNC Build V3.00

P-CHAN-00262	**Define kinematic ID for transformations**
Description	The kinematic ID identifies the related transformation as an element of the data set of kinematic parameters. The definition can be made for single-step, multi-step and PCS transformations.
Parameter	trafo[j].id kin_step[i].trafo[j].id (multi-step transformations) trafo_pcs[i].id (PCS transformation *)
Data type	UNS16
Data range	1 ... MAX(UNS16)
Dimension	----
Default value	0
Remarks	Parameter syntax as of V300 and higher *The PCS transformation function is available as of V3.1.3110.

P-CHAN-00263	Define kinematic parameters for multi-step transformations
Description	The specific kinematic offsets are entered in this structure for each transformation. You can specify offsets for single-level, multilevel transformations and PCS transformations.
Parameter	trafo[j].param[k] or where k = 0 to 73 (maximum number of kinematic parameters) kin_step[i].trafo[j].param[k] (multi-step transformations) trafo_pcs[i].param[k] (PCS transformation *)
Data type	REAL64
Data range	----
Dimension	0.1 µm or 0.0001 inch
Default value	0
Remarks	Kinematic parameters can also be entered in the tool data list P-TOOL-00009 (they are then relevant only when a tool is selected, independent of the kinematics). If a kinematic parameter is assigned in both lists, the specified values are added in the NC. This is only valid for transformation step 1. No additional kinematic parameters can be entered in the tool data for transformation step 2. For further details on the parameterisation of a kinematic transformation, see [KITRA] and [PROG]. (Parameter syntax as of V300) *The PCS transformation function is available as of V3.1.3110.

For CNC Builds < V3.00

P-CHAN-00094	Define kinematic parameters
Description	The specific kinematic offsets are entered in this structure for each transformation.
Parameter	kinematik[i].param[j] where j = 0... 74 (maximum number of kinematic parameters, application-specific, syntax as of V2.10.1501)
Data type	REAL64
Data range	----
Dimension	0.1µm or 0.0001° (for offsets)
Default value	0
Remarks	**Parametrising kinematics for CNC Builds V2.11.2xxx** *kinematik[i].wz_kopf_versatz[j] (syntax up to V260)* Kinematic parameters can also be entered in the tool data list P-TOOL-00009 (they are then relevant only when a tool is selected independent on the kinematics). If a kinematic parameter is assigned in both lists, the specified values are added in the NC. For more details on parameterising kinematic transformation and 5-axis machining, see [KITRA] and [PROG]. Parameterisation example: Offset values are specified for kinematics 1, 2 and 5. The kinematic with ID 2 is the default kinematic. `kinematik_id 2 Default kinematic: 2` `#` `kinematik[1].param[0] 1088000` `kinematik[1].param[1] 1987000` `kinematik[1].param[2] 342000` `#` `kinematik[2].param[0] 1538000` `kinematik[2].param[1] 25000` `kinematik[2].param[2] 0` `kinematik[2].param[5] 1800000` `#` `kinematik[5].param[0] 1487000` `kinematik[5].param[1] 25000`

3.2.1.2.1 Tool offsets

The commands #FACE and #CYL result in the implicit selection of kinematics. For this reason, neither a kinematic ID requires a #KIN ID [..] nor transformation activation with #TRAFO ON.

Tool offsets for face machining

Face machining supports 2 machine types (lathe/milling machine). The corresponding tool offsets must be entered in the channel parameters in the assigned offset data of the kinematic IDs 13 and 14. Alternatively, this can also be executed in the tool data.

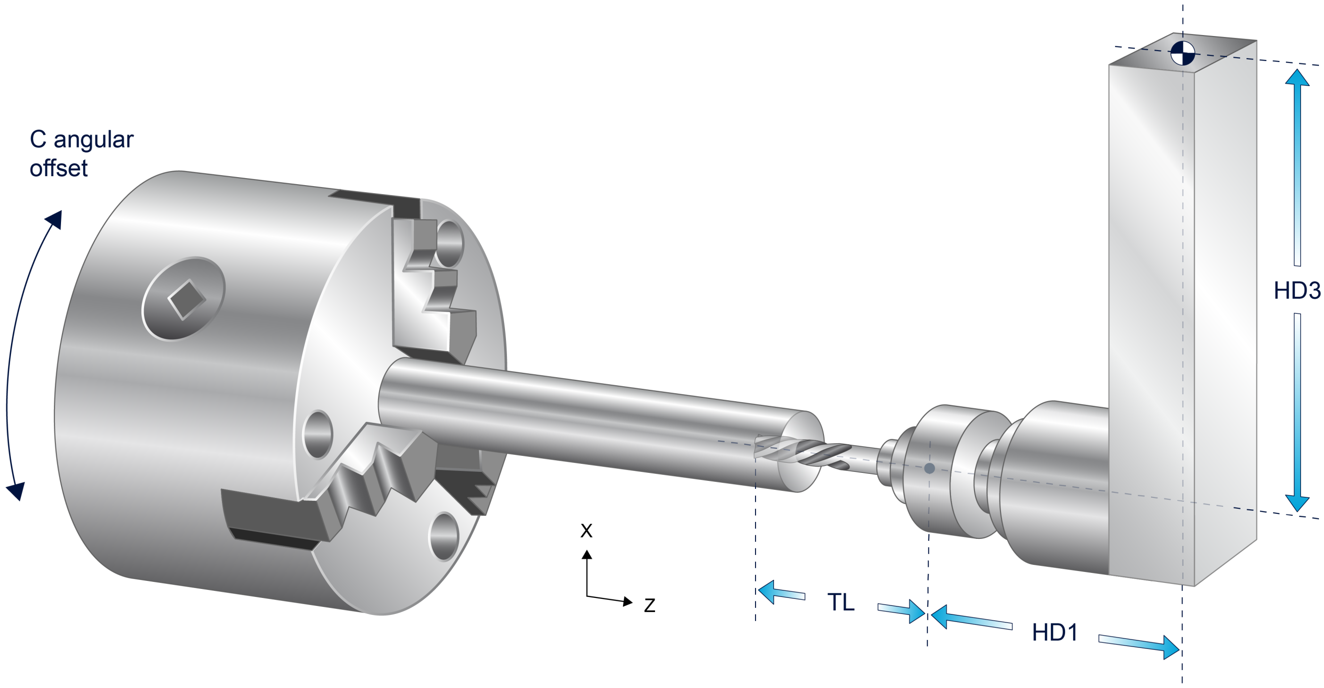


Fig. 20: Tool offsets for face machining

Example

Examples of entries in channel parameters

for CNC Builds as of V3.00

FACE[], Face machining on a <u>lathe</u> (KIN-ID 13):

```
trafo[0].id                     13
trafo[0].param[0]               1080000   Z offset [0.1µm]
trafo[0].param[1]               0        C angular offset [10^-4°]
trafo[0].param[2]               900000   X offset [0.1µm]
```

FACE[], Face machining on a <u>milling machine</u> (KIN-ID 14):

```
trafo[1].id                     14
trafo[0].param[0]               1080000   Z offset [0.1µm]
trafo[0].param[1]               0        C angular offset [10^-4°]
trafo[1].param[2]               900000   X offset [0.1µm]
```

CNC Builds < V3.00

FACE[], Face machining on a <u>lathe</u> (KIN-ID 13):

```
kinematik[13].param[0]          1080000   Z offset [0.1µm]
kinematik[13].param[1]           0        C angular offset [10^-4°]
kinematik[13].param[2]           900000   X offset [0.1µm]
```

FACE[], Face machining on a <u>milling machine</u> (KIN-ID 14):

```
kinematik[14].param[0]          1080000  Z offset [0.1µm]
kinematik[14].param[1]          0        C angular offset [10^-4°]
kinematik[14].param[2]          900000   X offset [0.1µm]
```

Tool offsets for lateral surface machining

Lateral surface machining implicitly executes the selection of the kinematic with ID 15.

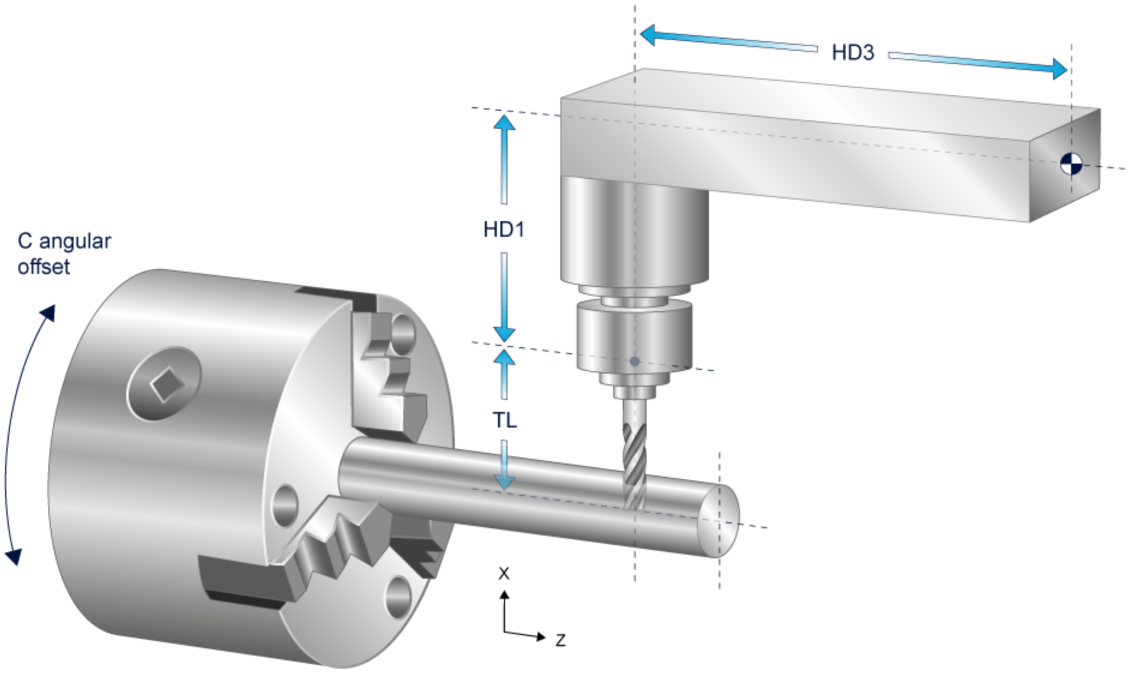


Fig. 21: Tool offsets for lateral surface machining

#CYCL[], Lateral surface machining lathe (KIN-ID 15):

CNC Builds as of V3.00

```
trafo[0].id                   15
trafo[0].param[0]             700000    X offset [0.1µm]
trafo[0].param[0]             0         C angular offset [10-4°]
trafo[2].param[0]             1200000    Z offset [0.1µm]
```

CNC Builds < V3.00

```
kinematik[15].param[0]        700000    X offset [0.1µm]
kinematik[15].param[1]        0         C angular offset [10-4°]
kinematik[15].param[2]        1200000   Z offset [0.1µm]
```

3.2.2 Axis parameter

Overview

The specific parameters for spindle axes and linear axes are described here. Refer to the documentation [AXIS] for details of further parameters available for setting axes.

Operation mode settings (P-AXIS-00015) for spindle axes

- 0x4: A modulo calculation always takes place after the destination position has been reached. Regardless of the operation mode selected for rotary axes, a modulo calculation is always executed in the position controller. In this way, modulo circle compensation can be executed if required.
- 0x100: For a spindle an automatic homing before positioning the spindle can be pre-vented. This is, of course, relevant only if the axis is not referenced. The function is drive-dependent.
- 0x200: Axis for kinematic "C axis" transformation

Operation mode settings (P-AXIS-00015) for path axes

- 0x1: Linear axis
- 0x40: Face turning axis
- 0x80: Longitudinal turning axis

P-AXIS-00015	**Axis mode**	
Description	Axes can be traversed in different operating modes.	
Parameter	kenngr.achs_mode	
Data type	UNS32	
Data range	0x00000001 - 0x10000000	
Axis types	T, R, S	
Dimension	T: ----	R,S: ----
Default value	0x00000001	
Drive types	----	
Remarks		

P-AXIS-00016	**Logical axis number**	
Description	The logical axis number is a system-wide unique identifier for each axis. The entire management of axis data in the NC kernel takes place using the logical axis number.	
Parameter	kopf.achs_nr	
Data type	UNS16	
Data range	0 < achs_nr < MAX(UNS16)	
Axis types	T, R, S	
Dimension	T: ----	R,S: ----
Default value	1	
drive types.	----	
Remarks	It is not allowed to use the same logical axis number more than once. The logical axis number "0" is not allowed. The assignment of an axis designation in the NC program to a logical axis (axis number) takes place in the channel parameters [CHAN]. This entry is not adopted when the axis parameter list is updated. Updates only become effective when the controller is rebooted.	

P-AXIS-00018	**Axis type (linear axis, rotary axes, spindle)**	
Description	This parameter specifies the axis type of an axis.	
Parameter	kenngr.achs_typ	
Data type	STRING	
Data range	Linear axis (ACHSTYP_TRANSLATOR) : 0x0001 Rotary axis (ACHSTYP_ROTATOR) : 0x0002 Spindle (ACHSTYP_SPINDEL): : 0x0004	
Axis types	T, R, S	
Dimension	T: ----	R,S: ----
Default value	ACHSTYP_TRANSLATOR	
Drive types	----	
Remarks	Depending on the axis type that is set, special functionalities are addressed in the NC kernel. Examples: - modulo calculation for rotary axes, - speed monitoring for spindles	

P-AXIS-00220	**Limiting velocity for the measurement system**	
Description	Especially with spindles, the limit at which the measuring system supplies erroneous signals at higher rotation speeds can be exceeded. The parameter describes the velocity value at which the position controller has to switch over to open-loop controlled operation mode.	
Parameter	getriebe[i].vb_regelgrenze	
Data type	UNS32	
Data range	0 ≤ vb_regelgrenze ≤ MAX(UNS32)	
Axis types	S	
Dimension		S: 0.001°/s
Default value	200000	
drive types.	Simulation, Conventional, Terminal, Lightbus, Profidrive	
Remarks		

P-AXIS-00216	**Limit for spindle speed 'zero'**	
Description	Especially in the case of spindles, the barrier should be given below which rotational speed monitoring in the position controller shows the state 'speed zero'.	
Parameter	getriebe[i].vb_min_null	
Data type	UNS32	
Data range	0 ≤ vb_min_null ≤ MAX(UNS32)	
Axis types	S	
Dimension		S: 0.001°/s
Default value	100	
drive types.	----	
Remarks		

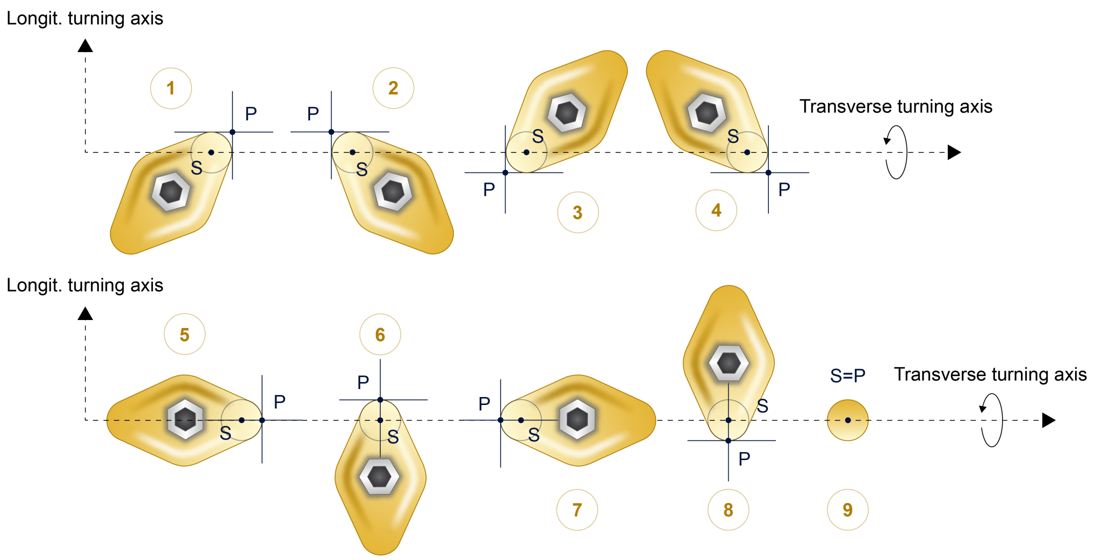
P-AXIS-00058	**Diameter programming absolute**	
Description	If diameter programming is to occur with active absolute programming (G90), the parameter must be set to TRUE.	
Parameter	kenngr.durchm_prog_abs	
Data type	BOOLEAN	
Data range	0/1	
Axis types	T	
Dimension	T: ----	
Default value	0	
drive types.	----	
Remarks		

P-AXIS-00059	Diameter programming relative	
Description	If diameter programming is to occur with active incremental programming (G91), the parameter must be set to TRUE.	
Parameter	kenngr.durchm_prog_rel	
Data type	BOOLEAN	
Data range	0/1	
Axis types	T	
Dimension	T: ----	
Default value	0	
drive types.	----	
Remarks		

3.2.3 Tool parameters

P-TOOL-00001	Type
Description	This parameter is assigned to distinguish between tool types.
Parameter	wz[i].typ or wz[i].type
Data type	UNS16
Data range	0: Milling tool 1: Turning tool 2: Grinding tool 3: Wire (erosion)
Dimension	----
Default value	0
Remarks	Parameterisation example: Tool 5 is a milling tool *wz[5].typ 0*

<table>
<tr><th>P-TOOL-00002</th><th>Cutter edge position</th></tr>
<tr><td>Description</td><td>With rotary tools (P-TOOL-00001 = 1), this parameter specifies the orientation of the cutter relative to the machining plane.



Ident. codes 1 to 9 for orientation of the turning tool tip in the machining plane.</td></tr>
<tr><td>Parameter</td><td>wz[i].srk_lage</td></tr>
<tr><td>Data type</td><td>UNS16</td></tr>
<tr><td>Data range</td><td>0 < srk_lage ≤ 9</td></tr>
<tr><td>Dimension</td><td>----</td></tr>
<tr><td>Default value</td><td>0</td></tr>
<tr><td>Remarks</td><td>For detailed information regarding machining with turning tools, see [PROG [▶ 12]].
Parameterisation example: The orientation of the cutter tip of Tool 6 is described by identification code 3.
wz[6].typ 1
wz[6].srk_lage 3</td></tr>
</table>

4 Appendix

4.1 Suggestions, corrections and the latest documentation

Did you find any errors? Do you have any suggestions or constructive criticism? Then please contact us at documentation@isg-stuttgart.de.

The latest documentation is posted in our Online Help (DE/EN):

QR code link: https://www.isg-stuttgart.de/documentation-kernel/

The link above forwards you to:

https://www.isg-stuttgart.de/fileadmin/kernel/kernel-html/index.html

Notice

Change options for favourite links in your browser;

Technical changes to the website layout concerning folder paths or a change in the HTML framework and therefore the link structure cannot be excluded.

We recommend you to save the above "QR code link" as your primary favourite link.

PDFs for download:

DE:
https://www.isg-stuttgart.de/produkte/softwareprodukte/isg-kernel/dokumente-und-downloads

EN:
https://www.isg-stuttgart.de/en/products/softwareproducts/isg-kernel/documents-and-downloads

Email: documentation@isg-stuttgart.de

Index

P

kernel