Rexroth
Bosch Group

Rexroth MTC 200
Graphic NC Programming

R911298712
Edition 01

Application Manual

Title

Rexroth MTC 200
Graphic NC Programming

Type of Documentation Application Manual

Document Typecode DOK-MTC200-GNP*****V23-AW01-EN-P

Internal File Reference Document Number 120-1701-B308-01/EN

Purpose of Documentation This documentation describes the Graphic NC Programming.

Record of Revisions

Description	Release Date	Notes
120-1701-B308-01/EN	12.2003	Valid from version 23

Published by Bosch Rexroth AG
Bgm.-Dr.-Nebel-Str. 2 • D-97816 Lohr a. Main
Telephone +49 (0)93 52/40-0 • Tx 68 94 21 • Fax +49 (0)93 52/40-48 85
http://www.boschrexroth.com/
Dept. BRC/ESM2 (JAD)
Dept. BRC/ESM6 (DiHa)

Note This document has been printed on chlorine-free bleached paper.

Contents

1 General Rules Regarding Graphic NC Programming

1.1 Preface

Function

Graphic NC programming (GNP) represents an efficient and highly precise tool that supports NC parts programming. It enables the user to easily define geometric elements, such as contours, and their machining.

GNP generates NC code in the syntax defined for the MTC 200 (see the NC programming instructions).

CAUTION

Due to the extremely large number of possible combinations between the geometry and the processing variance, the NC programmer must check and introduce the GNP results with the required care. A recommended aide for the check is NC simulation.

Ordering information

GNP is an optional component for the MTC 200 user interface and can be ordered under type key SWS-MTC200-GPT-23VRS-MS.

1.2 Conditions

License

Using GNP requires a corresponding license. The license can be ordered under Main Menu/Start Setup/License Management/Code.

Set compiler

Simultaneously, the compiler must be activated within the NC Editor in menu Extras/Compiler settings (place a checkmark in front of Compiler).

NC axes

The machine must have corresponding NC axes that allow turning/milling. This configuration must be reflected in the loaded set of parameters.

1.3 General Notes

Application

Using GNP, it is possible to program typical machining duties such as the roughing of turning contours.

Instructions

GNP instructions are divided into two main groups:

- **Geometry definitions** determine the location of machining (e.g. contours). They have only the defined character within the NC program.
- **Machining cycles** are technology subprograms with which the actual machining procedure is implemented (e.g. rotating, rough machining/finish machining).

GNP instructions can be described either by "normal" line editing (variant 1) within the NC Editor or by using an dialog box for entering/changing data (variant 2).

Variant 1

GNP parameters are entered/corrected using the NC Editor in a free format corresponding to the required syntax.

Variant 2

GNP is called within the NC Editor of the graphic user interface using dialog boxes. Use <F6> "Insert" to **insert new instructions**. In the following new F key level, you can select between either <F7> "GNP contour" or <F8> "GNP cycle" according to the desired instruction group. Then the NC Editor is switched to the graphic view. After returning to the

NC program view, the new contents are inserted in front of the current line position of the cursor.

Dialog box-guided **corrections** are to be made using <F7> "GNP correction". The cursor must first be positioned on the line or sequence of lines (contours) to be modified.

1.4 Implementation Level in this Version

Not all machining cycles have been implemented in the current version. The following have been implemented:

- G710 Turning, rough machining cycle
- G711 Turning, finish machining cycle

Machining cycles described in this documentation that are not yet available are dimmed.

2 Description of the Instructions

2.1 Geometry Definitions

Contours

Function

A workpiece drawing that is suitable for NC purposes must contain all the coordinates required for programming linear patterns and circles. Since drawings often show more constructional concerns than production-orientated ones, extensive mathematical calculations are sometimes required to determine the coordinates. Contour movement programming makes this significantly easier. It consists of header information, the starting point, a series of arcs, linear patterns, transition elements and an end instruction. The contour description itself does not trigger any movement. The corresponding movement blocks are generated only after the contour ID is called as a parameter of a machining cycle.

As a rule, the commands and parameters are effective on a block-by-block basis, i.e. constant parameters and G commands must also be reprogrammed in the subsequent block.

The given values in the relevant block are used to calculate an element. If these are not conclusive, the previous and subsequent elements are included in the calculation.

Position in NC program

The definition of a contour can be located anywhere within the NC program before its first application (e.g. in a cycle).

Header information

The "CONT" instruction is used to start the contour definition. The number of the contour is defined under "ID<1 - 1000>" as a required parameter. Additional parameters describe:

- radius/diameter programming (G15/G16)
- level (G17 - G22, G31, G32)
- axis designator (e.g. X, Y, Z)

If the default setting is used (see Configuration), the additional parameters are not required.

Example: Header information of contour number 100 in the default level:

CONT ID100

Starting point G100

The actual contour definition begins by specifying a starting point. The starting point instruction is defined by "G100" and the associated starting coordinates in the level. The addresses conform to the axis addresses of the CNC.

Example: The starting point of the contour (not of machining) begins in X20 and Z0.

G100 X20 Z0

Subsequent elements G101/102/103

Then the contour path is predefined by selecting subsequent elements. Subsequent elements are linear patterns and arcs.

G101 linear patterns:

A linear pattern is determined by:

- the end point coordinate(s) and/or
- the angle with the positive horizontal axis (**AA**) and/or
- the length (radius) (**R**) and/or
- the tangent transition to the previous element (**TG**) and/or
- the solution selection (**MD1/2**) (if there are several possible solutions).

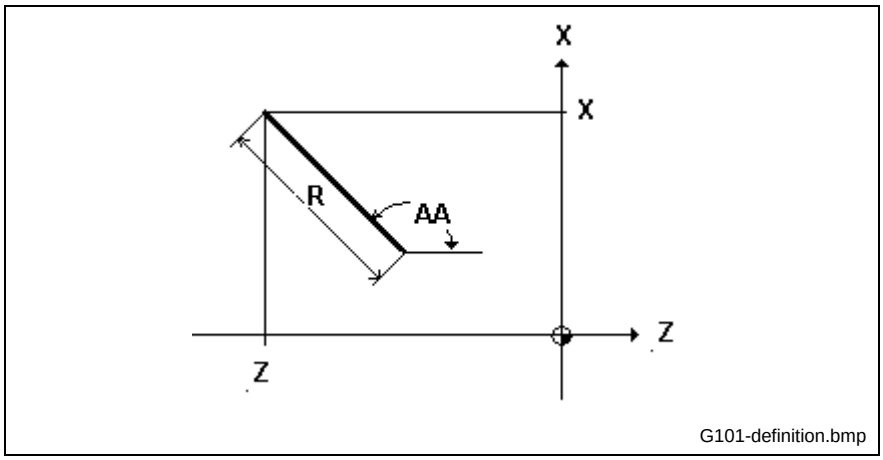


Fig. 2-1: Linear pattern

Example 1: A linear pattern is defined by an end point in X=100 and an angle of 135 degrees.

G101 X100 AA135

Example 2: A linear pattern is defined by an angle of 122 degrees and a linear pattern length of 140 (radius):

G101 AA122 R140

G102/G103 arcs:

An arc is determined by:

- the direction of the circle's sense of rotation: right = **G102**, left = **G103.**
- the absolute end point coordinate(s) and/or
- the starting angle of the arc with the positive horizontal axis (**AA**) and/or
- the end angle of the arc with the positive horizontal axis (**EA**) and/or
- the absolute coordinate(s) of the center of the circle and/or
- the radius (**R**) and/or
- the tangent transition to the previous element (**TG**) or the tangential reverse, i.e. pointed transitions (**TR**) and/or
- the solution selection (**MD1/2**, if there are several possible solutions).

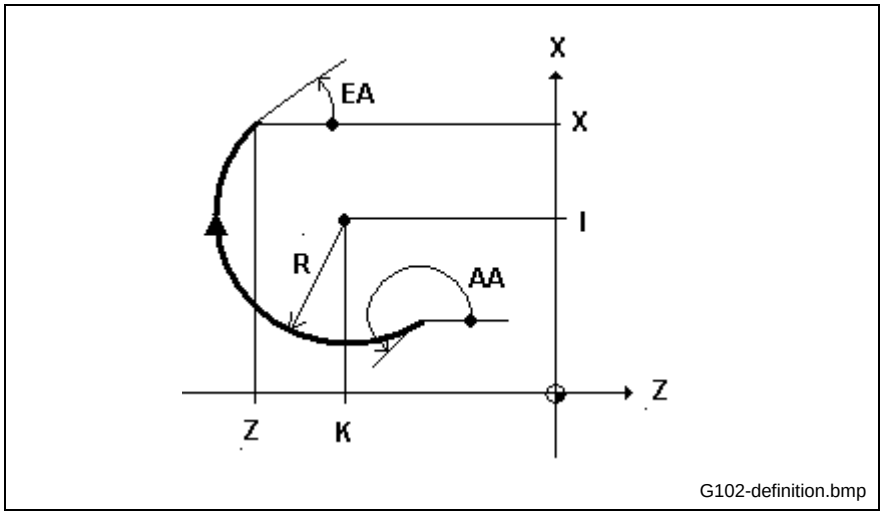


Fig. 2-2: Arc

Example: Arc to the right, center in I=23 in K=28, radius=16, end angle=45 degrees

G102 I0 K-28 R16 EA45

Alternative selection

Due to the range of parameters, there may be several mathematical solutions in the definition of contour movements. The alternative solutions are differentiated according to the following criteria:

Modifier **MD1** (default assumption) is used for each first assumption - otherwise, **MD2** is programmed.

Angle/length

- **Angle criterion:**
  - Solution selection → smaller angle = **"MD1"** - corresponds to the default setting (not required).
  - Solution selection → larger angle = **"MD2"** (must be programmed).

Note: The alternative selection must be programmed in the block in which the linear pattern is programmed.

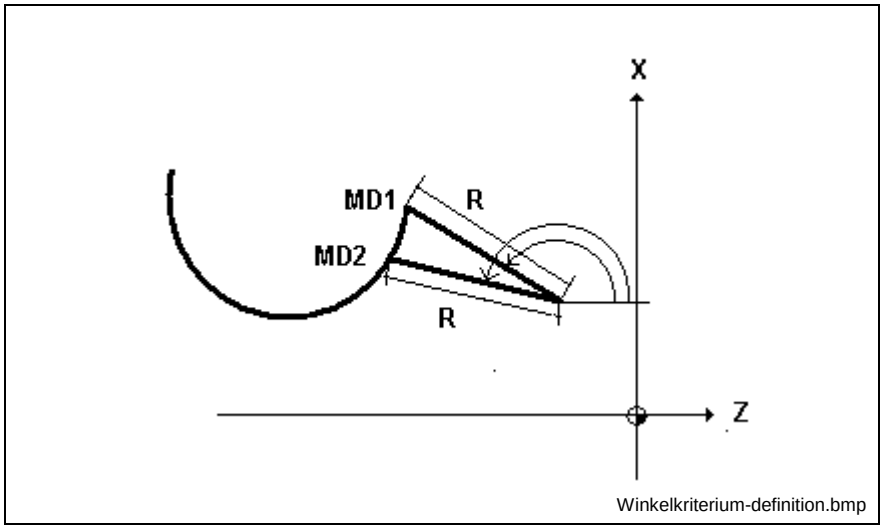


Fig. 2-3: Linear pattern - circle angle criterion

If the angle criterion results in no alternative (same angle), the length criterion is to be used.

- **Length criterion:**
  - Solution selection → shorter linear pattern = **"MD1"** - corresponds to the default setting (not required).
  - Solution selection → longer linear pattern = **"MD2"** (must be programmed).

Note: The alternative selection must be programmed in the block in which the linear pattern is programmed.

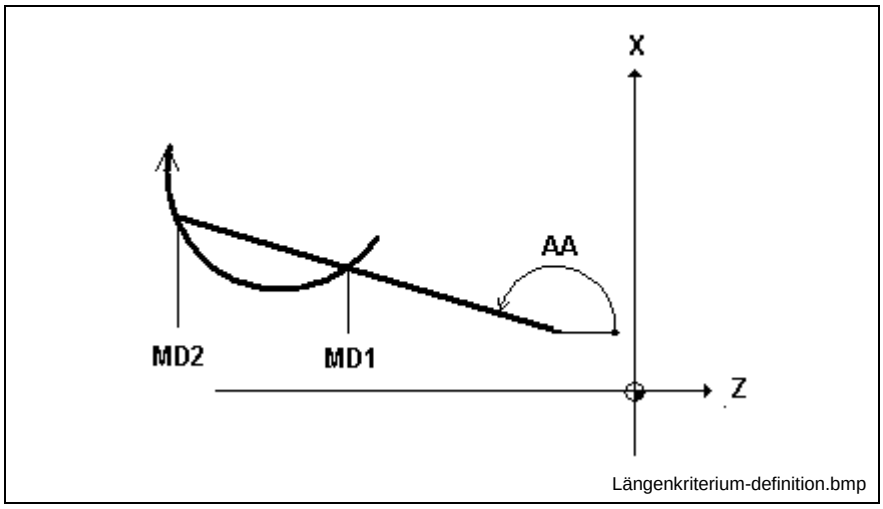


Fig. 2-4: Linear pattern - circle length criterion

Arc

- **Arc criterion:**
  - Solution selection → smaller arc = **"MD1"** - corresponds to the default setting (not required).
  - Solution selection → larger arc = **"MD2"** (must be programmed).

A contour movement consisting of an arc and a linear pattern is to be used as an example.

Description:

- The length and radius of the arc are specified by center coordinates I and K and starting point "P0".
- The contour end point is provided by end point coordinates X and Z.
- The starting point of the linear pattern lies on a circle with radius R.

Note: The alternative selection must be programmed in the block in which the arc is programmed.

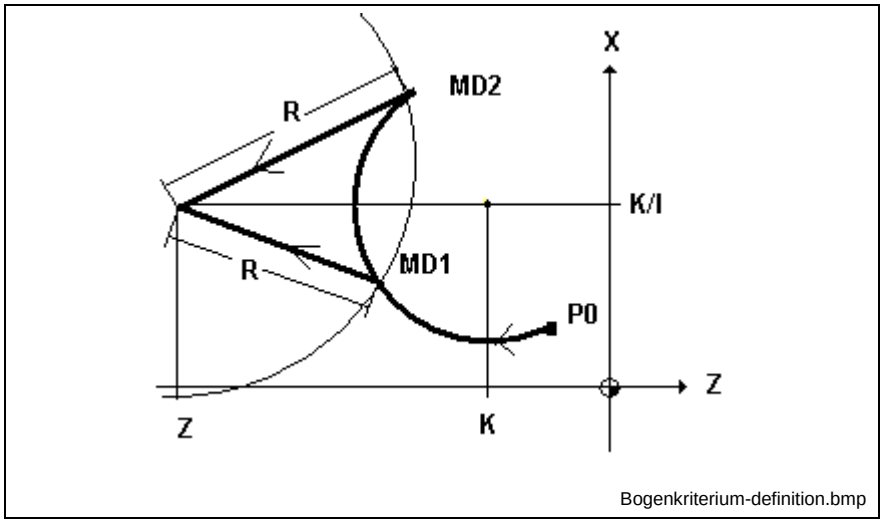


Fig. 2-5: Circle - linear pattern arc criterion

Tangential transitions

Depending on the range of parameters, there may be several solutions for tangential transitions between contour elements. The first solution **MD1** results if the tangent is aligned to the circle in the direction of the circle's orientation. Otherwise (the circle is aligned against the tangent), **MD2** must be programmed.

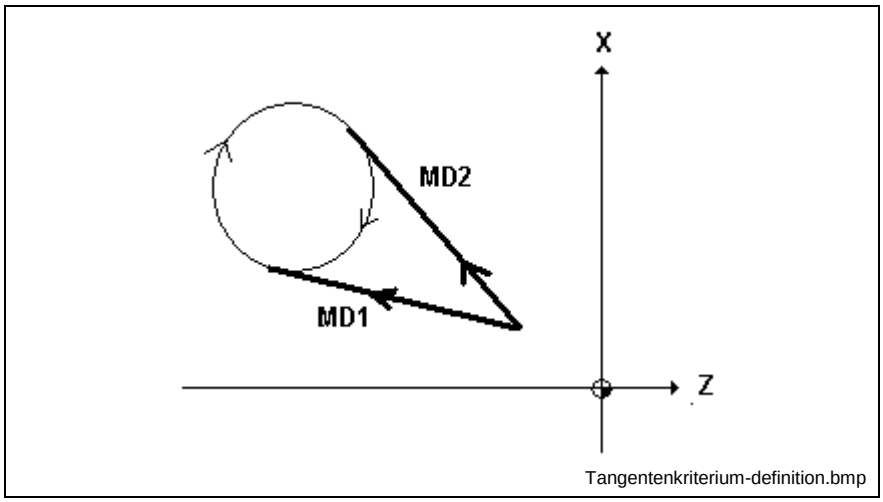


Fig. 2-6: Linear pattern - circle tangent criterion

Transition elements

Transition elements are elements between linear patterns and arcs. They are classified as follows:

Fillet radii

The linear pattern and arc elements can be filleted in any combination. They must only cut or contact one another. The fillet is programmed with:

G111 RD{value}

Sample fillet between two linear patterns with a radius of 0.5mm:

G101 X.....

G111 RD0.5

G101 X....

Chamfers

Symmetrical chamfers can be programmed between two subsequent linear patterns.

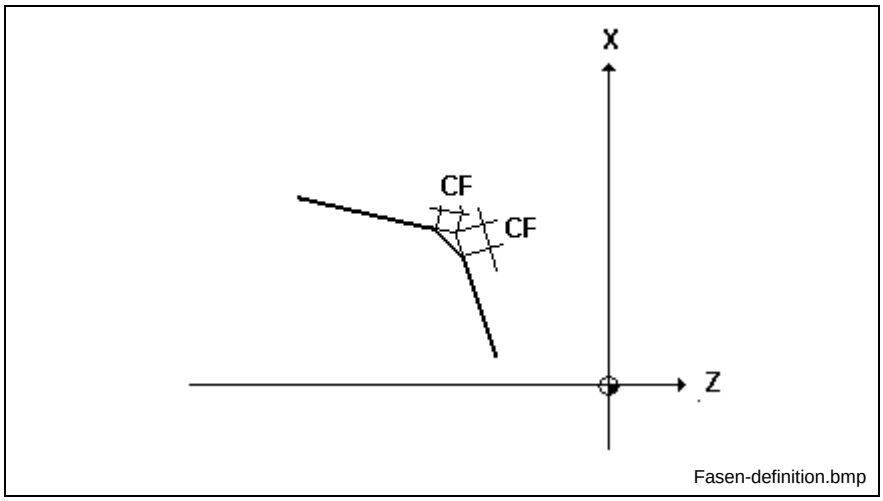


Fig. 2-7: Chamfer between linear patterns

The chamfer is programmed with:

G110 CF{value}

Sample chamfer between two linear patterns with a chamfer width of 5mm:

G101 X.....

G110 CF5

G101 X....

Undercut acc. to DIN509

Undercuts according to DIN 509 Form E and F can be programmed between two subsequent linear patterns.

Example: Undercut according to DIN 509 Form F:

G101 X.....

G112 RF F

G101 X....

End information

The "**END_CONT**" instruction is used to complete the contour definition.

Example

Description of a turning contour:

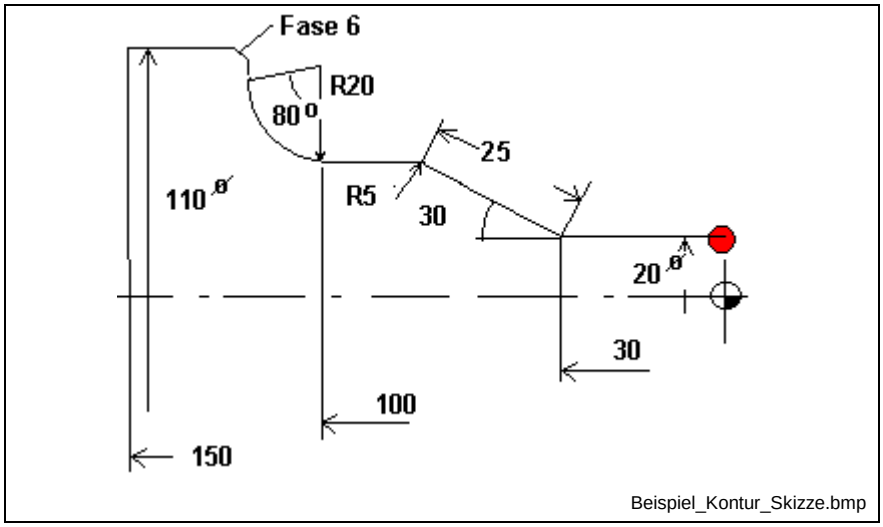


Fig. 2-8: Sample contour "Bolt"

```
CONT ID3              ; Name of contour "ID3"
G100 Z0 X20           ; Starting point of contour definition
G101 Z-30 X20         ; Linear pattern, X, Z
G101 AA150 L25        ; Radius, angle linear pattern
G111 RD5              ; Radius fillet
G101 Z-100 AA180      ; Linear pattern, Z, angle
G102 R20 EA100 TG     ; Circle, tangent, end angle, radius
G101 X110 TG          ; Linear pattern, tangent, X
G110 CF6              ; Chamfer, width
G101 Z-150 AA180      ; Linear pattern, Z, angle
END_CONT              ; End of contour definition
```

Mirror Imaging, Turning, Moving

Mirror imaging	Mirror imaging. Not yet implemented
Turning	Turning. Not yet implemented
Moving	Moving. Not yet implemented

Form Elements

Circle	Circle. Not yet implemented
Rectangle	Rectangle. Not yet implemented
Round groove	Round groove Not yet implemented
Straight groove	Straight groove. Not yet implemented
Polygon	Polygon. Not yet implemented
Straight text	Straight text. Not yet implemented

2.2 Machining Cycles

General Notes

Function Machining cycles are technology subroutines with which a specific machining procedure, such as Turning, rough machining or Turning, finish machining can be implemented. The specific machining sequence results from the parameters of the cycle.

Call A cycle is called by selecting the corresponding G code and the required parameter of the cycle.

Preconditions Before the cycle is called, the machining tool, the corresponding spindle speed / surface speed, the rotation direction and, if necessary, the gear range for the subsequent cycle must be programmed in the NC program.

Furthermore, a safe starting position from which positioning to the starting position of the movement sequence belonging to the cycle can be carried out without collisions must be approached before the cycle. At the end of the cycle, parameter "Tool retraction" can be used in various variants to return the position to the machining starting point before the cycle.

Return conditions At the end of the cycle, the conditions regarding the previously programmed G groups G15/G16, G90/G91 and G65/G94/G95 are the same as those before the cycle was called. In other words, they are reset internally at the end of the cycle.

Internal processing The actual NC control blocks that represent the cycle are generated in a compiler run and then inserted internally in the NC program at this point (compiled NC program).

Parameters All parameters have their own parameter address, allowing any sequence of parameters in the NC block. The value of a parameter can always be written immediately after the parameter address. Blank spaces to separate the parameter address and the value are permitted; they are required if the value starts with a letter. The decimal places are separated by a decimal point. Mathematical operations and variables are not allowed as parameter values. Macros, as a substitute for calling cycles/parameters, are also not permitted.

Required parameters are indicated by a "*". Functions that have not yet been implemented are "dimmed".

Note: It is not possible to describe all the combinations of parameters, including their effects, that are theoretically possible. Therefore, only those that occur most often in practice are discussed in the following description of parameters/cycles.

CAUTION

Due to the extremely large number of possible combinations between the geometry and the processing variance, the NC programmer must check and introduce the GNP results with the required care. A recommended aide for the check is NC simulation.

Default values for parameters

The parameters that are not required for the cycle and which were not specifically programmed are assigned default values at the start of cycle processing. Some of these default values can be modified using "Options"; others cannot be changed. Information regarding this is provided under "Default settings" within the parameter description.

DANGER

Danger of collisions due to copying NC programs from other processes/devices with possibly varying default parameter settings!

⇒ When transferring NC programs, check whether the cycles still are suitable to the possible different default settings.

G710 Turning, Rough Machining Cycle

Function

Using rough machining cycle G710, a previously defined contour can be machined in several cuts in a wide variety of technological working steps.

The cycle supports the machining of contours with a simple (single-edge) contour lathing tool.

G710 [Parameters]

Contour number ID*

Number of the contour that refers to the cycle. The contour, with the ID, must be defined previously in the subroutine.

Value range: 1 to 9999 (required parameter)

Cutting depth CD*

Maximum feed depth for every cut.

Value range: >0 (required parameter)

Feed F*

Machining feed F: basic measuring unit per rotation.

Value range: >0 (required parameter)

Allowance

Defines the size that must remain after machining for any further working steps on the contour.

Value range: >=0

A maximum of three different allowances can be programmed.

General allowance RSG

- General allowance. It applies to the entire contour if no additional programmed allowance for longitudinal or flat surfaces becomes effective for individual contour elements.

Flat allowance RSF

- Flat allowance; applies only to flat surfaces.

Longitudinal allowance RSL

- Longitudinal allowance; applies only to longitudinal surfaces (cylinder).

Note: Additional allowances for longitudinal and flat surfaces must be greater than general allowance RSG; otherwise, the general allowance is applied to these longitudinal/flat surfaces.

Under Options, it is possible to exclude linear patterns (contour elements), up to a certain length that can be input, from being subjected to an axis-parallel allowance. These guidelines can be defined separately for longitudinal and flat linear patterns in ascending contours and in undercuts. This setting can be used, for example, to attain the finishing of undercuts with simultaneous longitudinal allowance.

Tool retraction TRV

When cutting is complete, there are four tool retraction variants. They differ by the method that is used to determine whether and how the "Starting position" that was programmed **before** the cycle is to be attained at the end of the cycle. The default value can be specified under "Options":

- **TRV0** Tool does **not** travel to the starting position - it stops after the last step.
- **TRV1** Tool travels to the starting position **simultaneously** in both axes.
- **TRV2** Tool travels to the starting position **first** in the **flat axis** and then in the longitudinal axis.
- **TRV2** Tool travels to the starting position **first** in the **longitudinal axis** and then in the flat axis.

Cutting variant CV

The cutting variant specifies the technological sequence of cutting. The default setting can be made under "Options". A total of seven variants are available:

- **CV1** Axis-parallel, longitudinal, with contour tracing. After every step, the contour is traced until the "cutting step", which results from the cutting depth and the contour path, is also machined.
- **CV2** Axis-parallel, longitudinal.
- **CV3** Axis-parallel, flat, with contour cut. At the end of the axis-parallel cuts, a cut that eliminates the "cutting steps" is executed.
- **CV4** Contour-parallel cutting.
- **CV5** Axis-parallel, longitudinal, with contour cut. At the end of the axis-parallel cuts, a cut that eliminates the "cutting steps" is executed.
- **CV6** Axis-parallel, flat, with contour tracing. After every step, the contour is traced until the "cutting step", which results from the cutting depth and the contour path, is also machined.
- **CV7** Axis-parallel, flat.

Starting variant IFV

The starting variant specifies the relationship between the rough and the finished contour. Three variants are available:

- **IFV0** (default setting) - The blank results from the cylinder-shaped "wrapping" of the finished contour, including possible machining allowances.
- **IFV>0** The blank is specified by a contour-parallel machining allowance (e.g. forging).
- **IFV<0** The rough contour is specified by an additionally defined contour. The ID number of this contour, with a negative sign in front, is to be used as the value of the IFV parameter (see the following example).

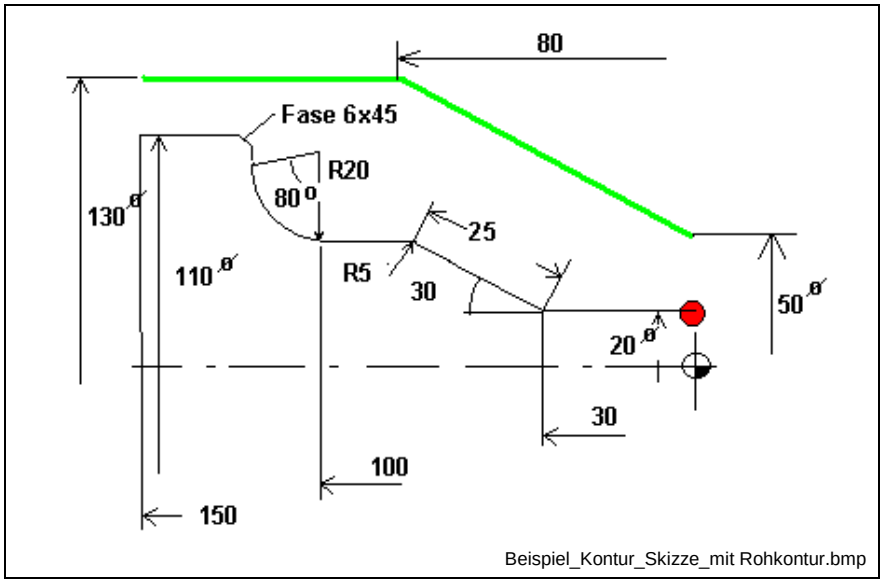


Fig. 2-9: "Bolt" contour with rough contour

Rough contour definition:

```
CONT ID30
 G100 X50 Z0
 G101 X130 Z-80
 G101 X130 Z-150
END_CONT
```

; Cycle section:

G710.....IFV-**30** ;With negative sign in front

Machining window The machining window limits/expands the machining space between the rough and the finished contours. In the case of extension, the machining contour and, if necessary, the rough contour are tangentially "lengthened" in the appropriate direction up to the window boundary. If a machining window is provided, all four values must always be programmed. The vertical window limits must be programmed in the radius/diameter unit, analog to the contour that is used.

Machining window WHT Top horizontal window limit.

Machining window WHD Bottom horizontal window limit.

Machining window WVR Right vertical window limit.

Machining window WVL Left vertical window limit.

Example:

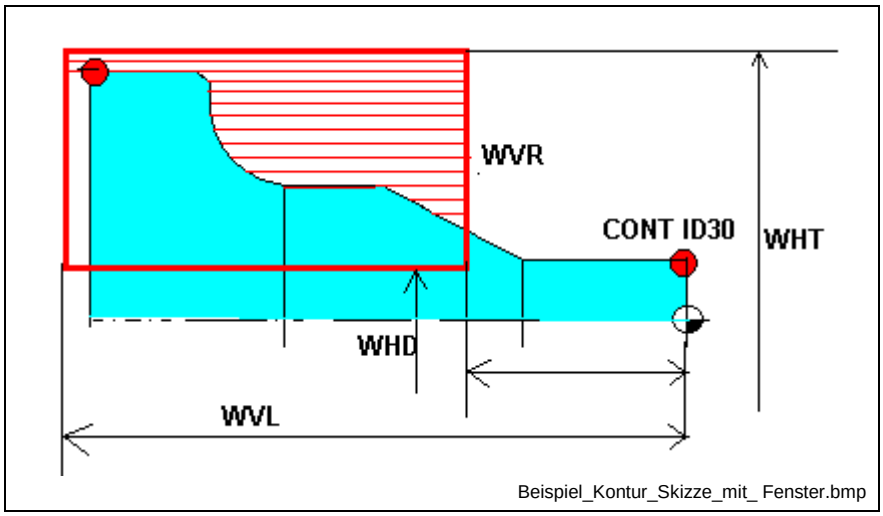


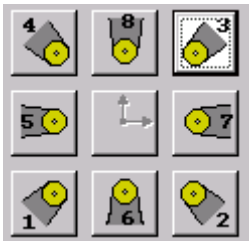
Fig. 2-10: Machining window

UCV undercut variants Specifies whether undercuts / rear cuts are to be included in machining.

- **UCV0** (default setting) - Machining is carried out **with** undercuts and **with** rear cuts.
- **UCV1** Machining is carried out **without** undercuts and **with** rear cuts.
- **UCV2** Machining is carried out **without** undercuts and **without** rear cuts.
- **UCV3** Machining is carried out **with** undercuts and **without** rear cuts.

Machining tool Tool data specify the geometry of the tool that is used for machining the cycle. Before executing the cycle, the tool edge position and the tool edge radius are checked in the CNC to see whether they still agree with the values at the generation of the cycle. If the values are not equal, a warning is displayed on the CNC. The start of the cycle must then be explicitly confirmed by the operator.

Tool edge position TEP Position of the tool edge acc. to the tool edge position code (default=3) of the CNC.



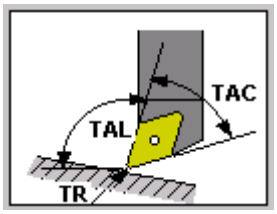
Schneidenlage.bmp

Fig. 2-11: Tool edge position code "behind center of rotation"

Tool radius TR Tool radius at the tool edge tip. The default setting can be made under "Options".

Tool setting angle TAL The basis for the specifications of the setting angle depends on the tool edge position. The angle is measured starting at the preferential cutting direction (axis-parallel) to the main tool edge. The default setting depends on the programmed tool edge position.

Tool corner angle TAC The corner angle is measured from the main tool edge to the secondary tool edge. The default setting depends on the programmed tool edge position.



Werkzeugwinkel_TEP3.bmp

Fig. 2-12: Tool angle for tool position 3

Lift tool variant RTV The lift tool variant determines whether the tool is lifted at the end of a cut or not.

- **RTV0** (default) - Lifting occurs at the end of every cut.
- **RTV1** - Lifting does **not** occur at the end of every cut.
- **RTV2** - Lifting **does** occur at the end of every cut - **except** for a contour cut.
- **RTV3** - Lifting does **not** occur at the end of every cut - **except** for a contour cut.

Chip breaking Parameter Path and Dwell Time can be used to determine how long and after which distance the tool is to come to a stop for the purpose of breaking a chip.

Chip breaking "distance" CBD Distance indication for which cutting is to be interrupted.

Chip breaking "time" CBT Time indication (seconds) of how long the cutting interruption should take.

Approach tangential AT

Using Approach tangential, the tool starting point can be influenced in addition to the machining window. The default value can be specified under "Options".

Approach normal* AN

Using Approach normal, the tool starting point can be influenced in addition to the machining window. *Not yet implemented.

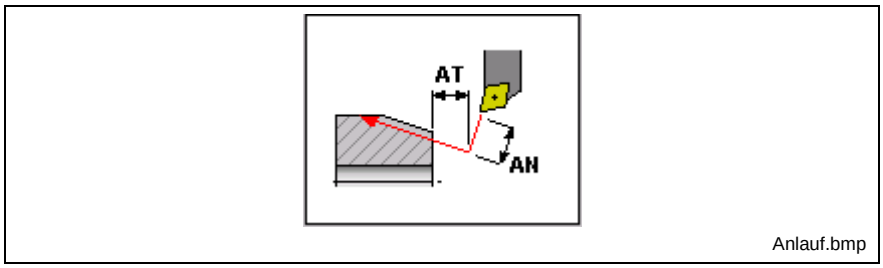


Fig. 2-13: Approach, tangential and normal

Runout variants RO

Using Runout variant RO, the machining end point can be influenced in addition to the machining window. The default value can be specified under "Options".

- **RO>0** - Distance that movement continues tangentially at the end of the contour or of the machining window (runout).
- **RO0** (default) - The tool continues to move by the amount of the tool edge radius at the end of the contour or of the machining window. The contour / machining window is machined without "remainders".
- **RO-1** - The end of the contour or the end limit of the machining window is **not** exceeded by the tool edge radius. This specification makes sense only for tools with tool edge position codes 1-4.

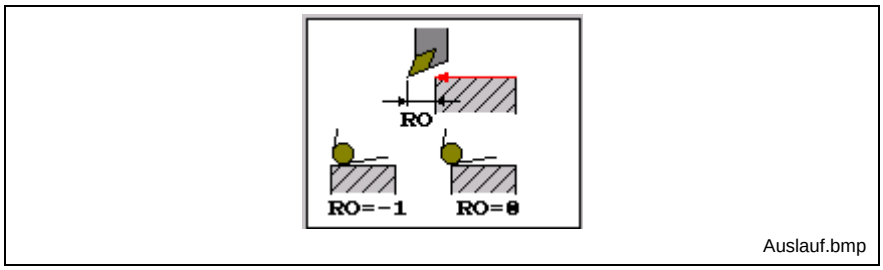


Fig. 2-14: Runout variants

G711 Turning, Finish Machining Cycle

Function

Using finish machining cycle G711, a previously defined contour can be machined using a one-time movement in a wide variety of technological variants.

The cycle supports the machining of contours with a simple (single-edge) contour lathing tool.

G711 [Parameters]

Contour number ID*

Number of the contour that refers to the cycle. The contour, with the ID, must have been defined previously in the subroutine.

Value range: 1 to 9999 (required parameter)

Feed F*

Machining feed F: basic measuring unit per rotation.

Value range: >0 (required parameter)

Allowance

Defines the size that must remain after machining for any further working steps on the contour.

Value range: >=0

A maximum of three different allowances can be programmed.

General allowance RSG

- General allowance. It applies to the entire contour if no additional programmed allowance for longitudinal or flat surfaces becomes effective for individual contour elements.

Flat allowance RSF

- Flat allowance; applies only to flat surfaces.

Longitudinal allowance RSL

- Longitudinal allowance; applies only to longitudinal surfaces (cylinder).

Note: Additional allowances for longitudinal and flat surfaces must be greater than general allowance RSG; otherwise, the general allowance is applied to these longitudinal/flat surfaces.

Under Options, it is possible to exclude linear patterns (contour elements), up to a certain length that can be input, from being subjected to an axis-parallel allowance. These guidelines can be defined separately for longitudinal and flat linear patterns in ascending contours and in undercuts. This setting can be used, for example, to attain the finishing of undercuts with simultaneous longitudinal allowance.

Tool retraction TRV

When cutting is complete, there are four tool retraction variants. They differ by the method that is used to determine whether and how the "Starting position" that was programmed **before** the cycle is to be attained at the end of the cycle. The default value can be specified under "Options":

- **TRV0** Tool does **not** travel to the starting position - it stops after the last step.
- **TRV1** Tool travels to the starting position simultaneously in both axes.
- **TRV2** Tool travels to the starting position first in the flat axis and then in the longitudinal axis.
- **TRV2** Tool travels to the starting position first in the longitudinal axis and then in the flat axis.

Machining window

The machining window limits/expands the machining space between the rough and the finished contours. In the case of extension, the machining contour and, if necessary, the rough contour are tangentially "lengthened" in the appropriate direction up to the window boundary. If a machining window is provided, all four values must always be programmed. The vertical window limits must be programmed in the radius/diameter unit, analog to the contour that is used.

Machining window WHT

Top horizontal window limit.

Machining window WHD

Bottom horizontal window limit.

Machining window WVR

Right vertical window limit.

Machining window WVL

Left vertical window limit.

Example:

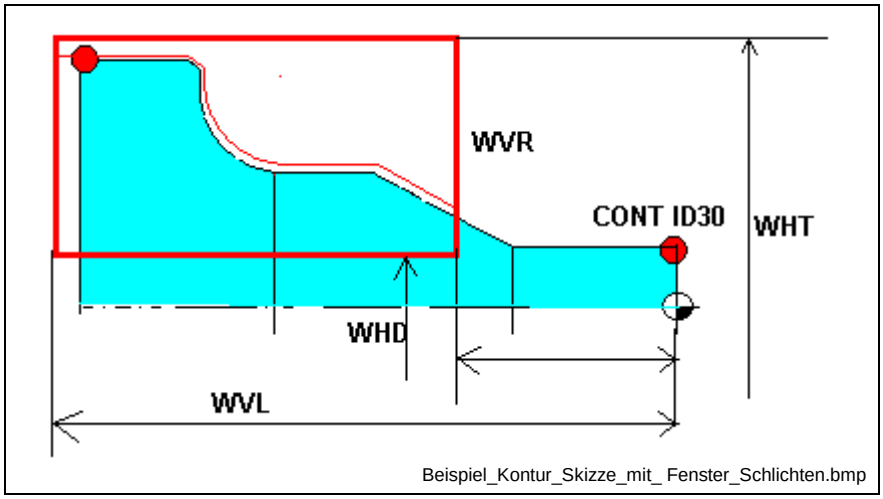


Fig. 2-15: Finish machining window

Machining tool

Tool data specify the geometry of the tool that is used for machining the cycle. Before executing the cycle, the tool edge position and the tool edge radius are checked in the CNC to see whether they still agree with the values at the generation of the cycle. If the values are not equal, a warning is displayed on the CNC. The start of the cycle must then be explicitly confirmed by the operator.

Tool edge position TEP

Position of the tool edge acc. to the tool edge position code (default=3) of the CNC.

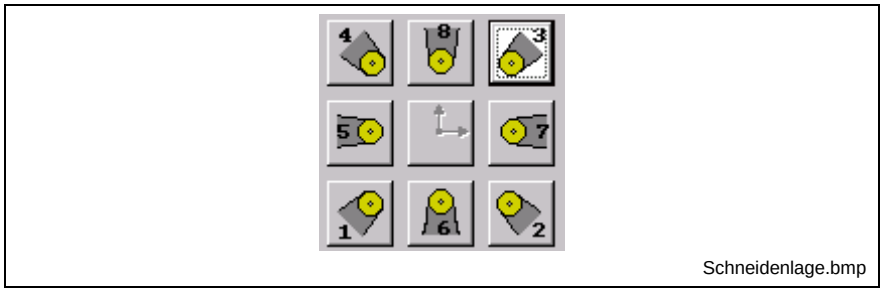


Fig. 2-16: Tool edge position code "behind center of rotation"

Tool radius TR

Tool radius at the tool edge tip. The default value can be specified.

Tool setting angle TAL

The basis for the specifications of the setting angle depends on the tool edge position. The angle is measured starting at the preferential cutting direction (axis-parallel) to the main tool edge. The default setting depends on the programmed tool edge position.

Tool corner angle TAC

The corner angle is measured from the main tool edge to the secondary tool edge. The default setting depends on the programmed tool edge position.

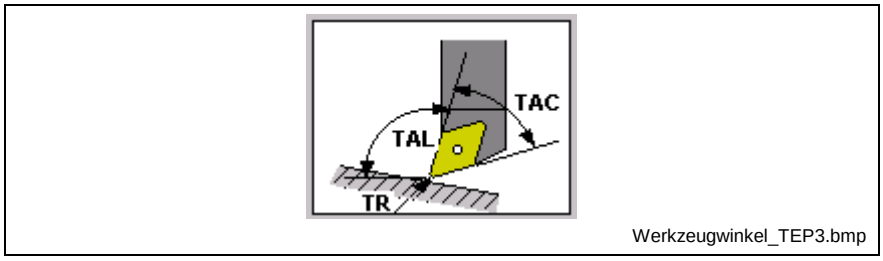


Fig. 2-17: Tool angle for tool position 3

Lift tool variant RTV

The lift tool variant determines whether the tool is lifted at the end of a cut or not.

- **RTV0** (default) - Lifting occurs at the end of every cut.
- **RTV1** - Lifting does **not** occur at the end of every cut.
- **RTV2** - Lifting **does** occur at the end of every cut - **except** for a contour cut.
- **RTV3** - Lifting does **not** occur at the end of every cut - **except** for a contour cut.

Approach tangential AT

Using Approach tangential, the tool starting point can be influenced in addition to the machining window. The default value can be specified.

Approach normal* AN

Using Approach normal, the tool starting point can be influenced in addition to the machining window. *Not yet implemented.

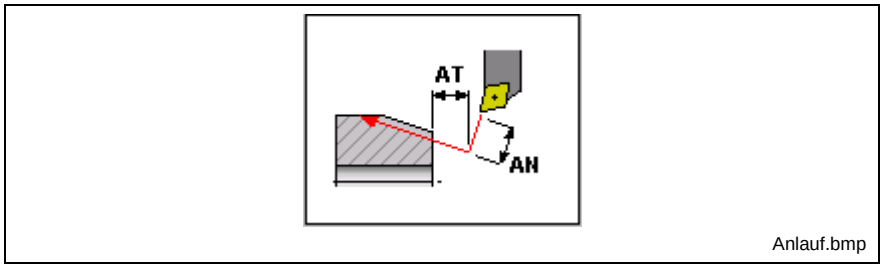


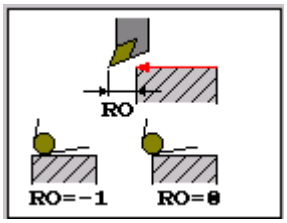
Fig. 2-18: Approach, tangential and normal

Runout variants RO

Using Approach, the tool end point can be influenced in addition to the machining window. The default value can be specified.

Using Runout variant RO, the machining end point can be influenced in addition to the machining window. The default value can be specified under "Options".

- **RO>0** - Distance that movement continues tangentially at the end of the contour or of the machining window (runout).
- **RO0** (default) - The tool continues to move by the amount of the tool edge radius at the end of the contour or of the machining window. The contour / machining window is machined without "remainders".
- **RO-1** - The end of the contour or the end limit of the machining window is **not** exceeded by the tool edge radius. This specification makes sense only for tools with tool edge position codes 1-4.



Auslauf.bmp

Fig. 2-19: Runout variants

G720 Plunging, Rough Machining Cycle

Function

Using rough machining cycle G720, a previously defined contour can be machined in several cuts in a wide variety of technological working steps.

The cycle especially supports the machining of form plunges with plunging tools (dual-edge parting tool and single-edge round-nose tool).

G720 [Parameters]

Contour number ID*

Number of the contour that refers to the cycle. The contour, with the ID, must be defined previously in the subroutine.

Value range: 1 to 9999 (required parameter)

Cutting depth CD

Maximum feed depth for every cut. The default is as follows for CV-1xx (plunging): the cutting depth is limited only by the tool length (TL).

The default is as follows for CV 2xx (rough machining): cutting depth = 0.7x (tool edge width)

Feed F*

Machining feed F: basic measuring unit per rotation.

Value range: >0 (required parameter)

Allowance

Defines the size that must remain after machining for any further working steps on the contour.

Value range: >=0

A maximum of three different allowances can be programmed.

General allowance RSG

- General allowance. It applies to the entire contour if no additional programmed allowance for longitudinal or flat surfaces becomes effective for individual contour elements.

Flat allowance RSF

- Flat allowance; applies only to flat surfaces.

Longitudinal allowance RSL

- Longitudinal allowance; applies only to longitudinal surfaces (cylinder).

Note: Additional allowances for longitudinal and flat surfaces must be greater than general allowance RSG; otherwise, the general allowance is applied to these longitudinal/flat surfaces.

Under Options, it is possible to exclude linear patterns (contour elements), up to a certain length that can be input, from being subjected to an axis-parallel allowance. These guidelines can be defined separately for longitudinal and flat linear patterns in ascending contours and in undercuts.

Tool retraction TRV

When cutting is complete, there are four tool retraction variants. They differ by the method that is used to determine whether and how the "Starting position" that was programmed **before** the cycle is to be attained at the end of the cycle. The default value can be specified under "Options":

- **TRV0** Tool does **not** travel to the starting position - it stops after the last step.
- **TRV1** Tool travels to the starting position simultaneously in both axes.
- **TRV2** Tool travels to the starting position first in the flat axis and then in the longitudinal axis.
- **TRV2** Tool travels to the starting position first in the longitudinal axis and then in the flat axis.

Cutting variant CV

The cutting variant specifies the technological sequence of cutting. A total of eight variants are available:

- **CV100** Plunging only, in one machining direction.

- **CV110** Plunging only, alternating machining directions.
- **CV111** Plunging only, alternating machining directions, contour cut.
- **CV101** (default) Plunging in one machining direction, contour cut.
- **CV200** Rough machining (plunging to deep CD, axis-parallel "Turning"), one machining direction.
- **CV210** Rough machining (plunging to deep CD, axis-parallel "Turning"), alternating machining directions.
- **CV211** Rough machining (plunging to deep CD, axis-parallel "Turning"), alternating machining directions, contour cut.
- **CV201** Rough machining (plunging to deep CD, axis-parallel "Turning"), one machining direction, contour cut.

Machining window The machining window limits/expands the machining space between the finished contour and the cylinder-shaped wrapping. In the case of extension, the machining contour is tangentially "lengthened" up to the window boundary. If a machining window is provided, all four values must always be programmed. The vertical window limits must be programmed in the radius/diameter unit, analog to the contour that is used.

Machining window WHT Top horizontal window limit.

Machining window WHD Bottom horizontal window limit.

Machining window WVR Right vertical window limit.

Machining window WVL Left vertical window limit.

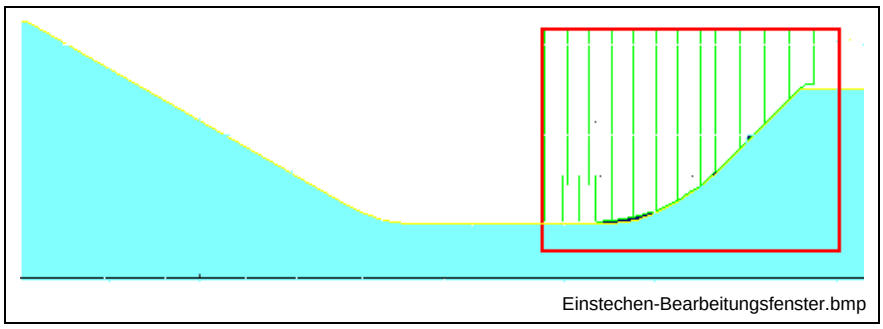


Fig. 2-20: Rough machining cycle machining window

UCV undercut variants This parameter specifies whether all "valleys" are to be completely machined together or one "valley" at a time.

- **UCV0** Machine all "valleys" together.
- **UCV1** Completely machine every "valley" individually.

Machining tool Tool data specify the geometry of the tool that is used for machining the cycle. Before executing the cycle, the orientation of the tool (radial/axial, inside/outside/right/left) and the tool edge width (tool edge radius in the case of a round-nose tool) are checked in the CNC to see whether they still agree with the values at the generation of the cycle. If the values are not equal, a warning is displayed on the CNC. The start of the cycle must then be explicitly confirmed by the operator.

Tool orientation TEP Orientation of the tool (default = 8, outside, radial). If a parting tool is used, the tool edge position code is to be programmed under TEP as for a round-nose tool with the same orientation. If a round-nose tool is used, its actual tool edge position code is to be specified.

Value range: 5 to 8

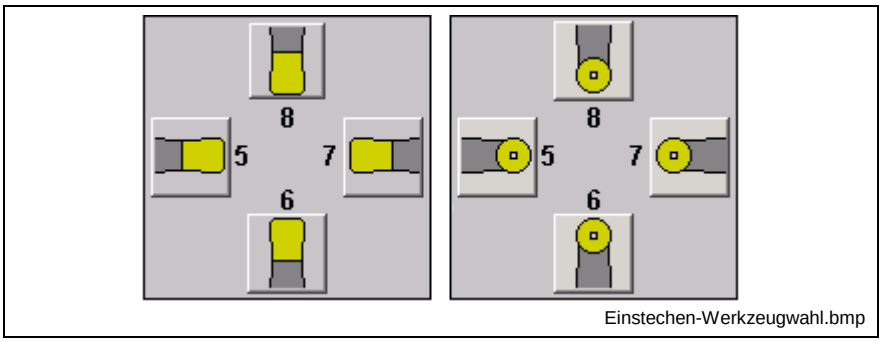


Fig. 2-21: Tool orientation "behind center of rotation"

Tool radius TR Tool edge radius. For dual-edge plunging tools, the radius applies to both tool edge corners. The default setting is defined under "Options".

Tool width TW* Width of the recessing tool (required parameter). "TW-1" is to be programmed for a round-nose tool.

Tool edge length TL Length of the tool edge. Standard:4 x TW, for TW= -1 → 8 x TR.

Tool edge number "A" TEA *Not yet implemented.

TEA is the number of the left/radial outer tool edge corner. It is called for recessing operations and - for contour cuts - the left (for TEP6+8) / radial outer (for TEP5+7) contour flank.

TEA must be specified for dual-edge parting tools if a contour cut occurs in the cycle (CV111, CV101, CV211, CV201).

Tool edge number "B" TEB *Not yet implemented.

Required only for contour cuts (CV111, CV101, CV211, CV201). Number of the right (for TEP6+8) / radial inner (for TEP5+7) tool edge. This is called for the right/radial inner contour flank in the case of contour cuts. TEB can not be specified for round-nose tools.

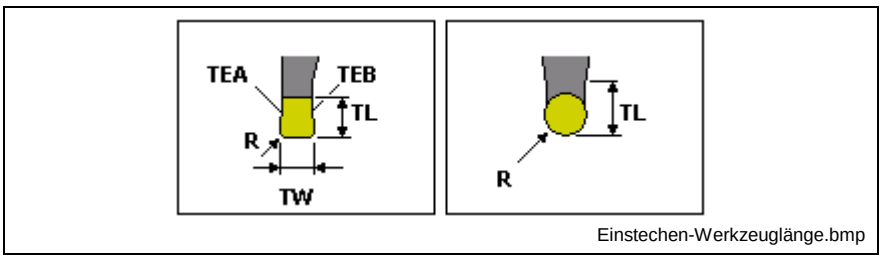


Fig. 2-22: Groove plunge tool data

Chip breaking Parameter Path and Dwell Time can be used to determine how long and after which distance the tool is to come to a stop for the purpose of breaking a chip.

Chip breaking "distance" CBD Distance indication for which cutting is to be interrupted.

Chip breaking "time" CBT Time indication (seconds) of how long the cutting interruption should take.

Approach tangential AT Using Approach tangential, the tool starting point can be influenced for every cut in addition to the machining window. The default value can be specified under "Options".

Starting point SP Machining starting point X or Z (depending on TEP) at which machining is to start.

Action conditions CG Action condition for plunging, in percent (100 = without lap). Standard 70%.

Machining direction CR This parameter can be used to switch the machining direction. CR0 means that the machining direction should be maintained, CR1 that it should be switched.

G721 Plunging, Finish Machining Cycle

Function Using finish machining cycle G721, a previously defined contour can be finished using recessing/round-nose tools.

G721 [Parameters]

Contour number ID* Number of the contour that refers to the cycle. The contour, with the ID, must be defined previously in the subroutine.

Value range: 1 to 9999 (required parameter)

Feed F* Machining feed F: basic measuring unit per rotation.

Value range: >0 (required parameter)

Allowance Defines the size that must remain after machining for any further working steps on the contour.

Value range: >=0

A maximum of three different allowances can be programmed.

General allowance RSG
- General allowance. It applies to the entire contour if no additional programmed allowance for longitudinal or flat surfaces becomes effective for individual contour elements.

Flat allowance RSF
- Flat allowance; applies only to flat surfaces.

Longitudinal allowance RSL
- Longitudinal allowance; applies only to longitudinal surfaces (cylinder).

Note: Additional allowances for longitudinal and flat surfaces must be greater than general allowance RSG; otherwise, the general allowance is applied to these longitudinal/flat surfaces.

Under Options, it is possible to exclude linear patterns (contour elements), up to a certain length that can be input, from being subjected to an axis-parallel allowance. These guidelines can be defined separately for longitudinal and flat linear patterns in ascending contours and in undercuts.

Tool retraction TRV When cutting is complete, there are four tool retraction variants. They differ by the method that is used to determine whether and how the "Starting position" that was programmed **before** the cycle is to be attained at the end of the cycle. The default value can be specified under "Options":

- **TRV0** Tool does **not** travel to the starting position - it stops after the last step.
- **TRV1** Tool travels to the starting position simultaneously in both axes.
- **TRV2** Tool travels to the starting position first in the flat axis and then in the longitudinal axis.
- **TRV2** Tool travels to the starting position first in the longitudinal axis and then in the flat axis.

Machining window The machining window limits/expands the machining space between the finished contour and the cylinder-shaped wrapping. In the case of extension, the machining contour is tangentially "lengthened" up to the window boundary. If a machining window is provided, all four values must always be programmed. The vertical window limits must be programmed in the radius/diameter unit, analog to the contour that is used.

Machining window WHT Top horizontal window limit.

Machining window WHD Bottom horizontal window limit.

Machining window WVR Right vertical window limit.

Machining window WVL Left vertical window limit.

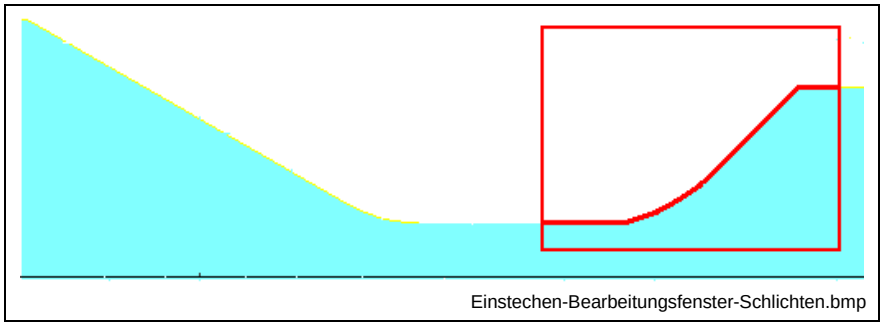


Fig. 2-23: Finish machining cycle machining window

Machining tool Tool data specify the geometry of the tool that is used for machining the cycle. Before executing the cycle, the orientation of the tool (radial/axial, inside/outside/right/left) and the tool edge width (tool edge radius in the case of a round-nose tool) are checked in the CNC to see whether they still agree with the values at the generation of the cycle. If the values are not equal, a warning is displayed on the CNC. The start of the cycle must then be explicitly confirmed by the operator.

Tool orientation TEP Orientation of the tool (default = 8, outside, radial). If a parting tool is used, the tool edge position code is to be programmed under TEP as for a round-nose tool with the same orientation. If a round-nose tool is used, its actual tool edge position code is to be specified.

Value range: 5 to 8

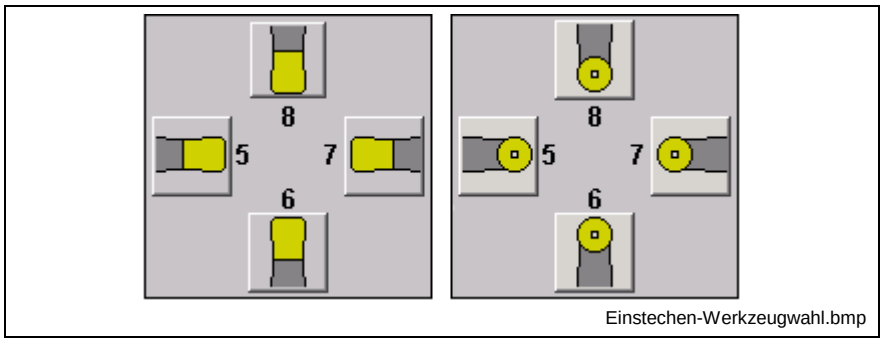


Fig. 2-24: Tool edge orientation "behind center of rotation"

Tool radius TR Tool edge radius. For dual-edge plunging tools, the radius applies to both tool edge corners. The default setting is defined under "Options".

Tool width TW Width of the recessing tool (required parameter). "TW-1" is to be programmed for a round-nose tool.

Tool edge length TL Length of the tool edge. Default: 4*TW, bei TW=-1 → 8*TR.

Tool edge number "A" TEA *Not yet implemented.

Number of the left/upper tool edge. It is called for the left (for TEP6+8) / radial outer (for TEP5+7) contour flank. Specification is not required for round-nose tools.

Tool edge number "B" TEB *Not yet implemented.

Number of the right (for TEP6+8) / radial inner (for TEP5+7) tool edge. This is called for the right/radial inner contour flank. Specification is not possible for round-nose tools.

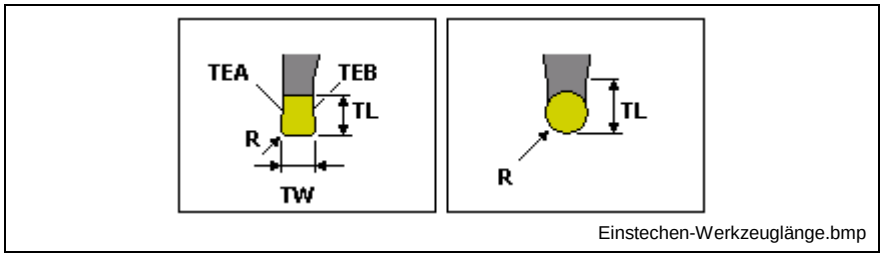


Fig. 2-25: Groove plunge tool data

Approach tangential AT Using Approach tangential, the tool starting point can be influenced for every cut in addition to the machining window. The default value can be specified under "Options".

Machining direction CR *Not yet implemented. This parameter can be used to switch the machining direction. CR0 means that the machining direction should be maintained, CR1 that it should be switched.

G782 Centric Drilling

Function

Drilling cycle G782 implements the entire movement sequence, including auxiliary movements, that is required for generating a centric hole.

Process:

In rapid traverse, the tool is positioned from the previously approached position to the center of rotation and then to safety distance "SD". From this point, drilling by the amount of chip depth "CD" proceeds with feed "F" (if CD is not programmed, movement to the final depth occurs immediately). A peck drilling motion "AD" is performed in rapid traverse after the feed. In the last step, drilling occurs to depth "DT"; at the end, dwell time "DWT" is run and then the tool is returned in rapid traverse to the load position.

G782 [Parameters]

Safety distance SD* This parameter specifies where drilling machining starts. This position is approached in rapid traverse. The dimension is absolute.

Depth DT* This parameter specifies where machining stops.

The dimension is absolute.

Feed F* Machining feed for the entire cycle.

Chip depth CD Feed depth - as an increment - for the first and every additional step until the final depth DT has been reached. If CD is not programmed, movement to the final depth occurs immediately.

Lift AD Lifting amount - as an increment - for chip breaking/removal.

Dwell time DWT Dwell time (seconds) at the end of machining.

Diameter D Diameter of the hole. Standard "0".

Axis group AGR Number of the axis group (default: default axis group).

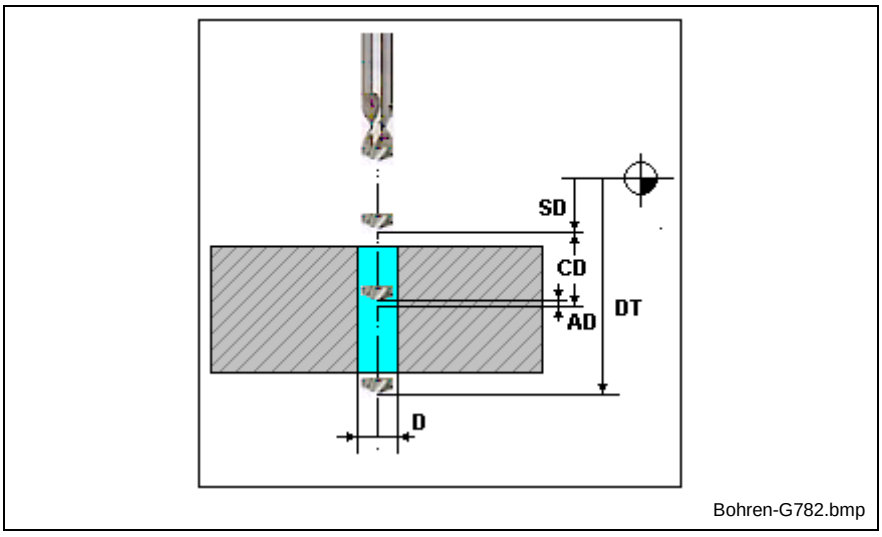


Fig. 2-26: G782 drilling

G787 Centric Teaming

Function Reaming cycle G787 implements the entire movement sequence, including auxiliary movements, that is required for machining a centric hole.

Process:

In rapid traverse, the tool is positioned from the previously approached position to the center of rotation and then to safety distance "SD". The next step is to ream to depth "DT" using feed "F" and then to let dwell time "DWT" elapse. Then the tool is positioned in feed "FR" to the safety distance and then returned in rapid traverse to the load position.

Safety distance SD* This parameter specifies where machining starts. This position is approached in rapid traverse. The dimension is absolute.

Depth DT* This parameter specifies where machining stops.

The dimension is absolute.

Feed F* Machining feed for plunging.

Feed retraction FR Feed for retraction. Standard value FR=2*F.

Dwell time DWT Dwell time (seconds) at the end of machining.

Diameter D Diameter of the hole. Standard "0".

Axis group AGR Number of the axis group (default: default axis group).

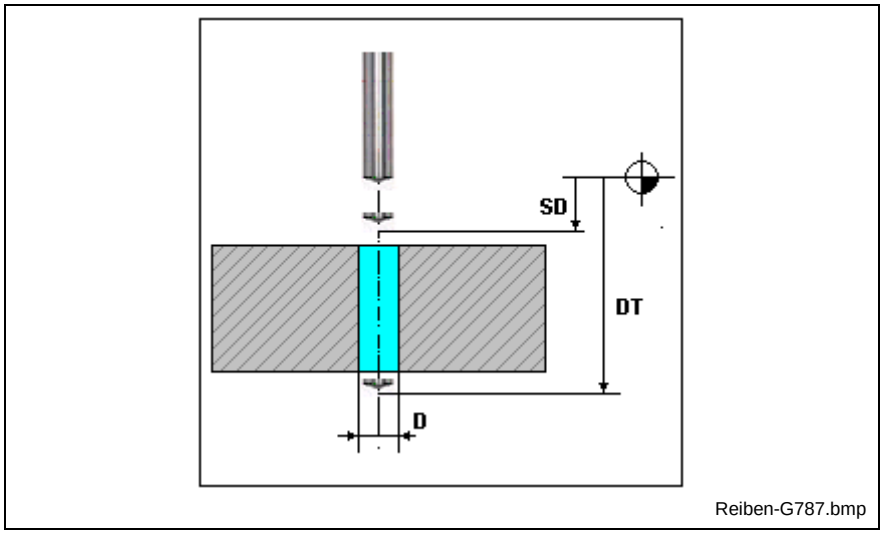


Fig. 2-27: G787 Reaming

G784 Centric Floating Tapping

Function

Floating tapping cycle G784 implements the entire movement sequence, including auxiliary movements, that is required for threading with a thread tapper.

Process:

In rapid traverse, the tool is positioned from the previously approached position to the center of rotation and then to safety distance "SD". The next step is to drill with feed to depth "DT" and then to reposition to safety distance "SD" in the feed with the spindle rotating in the opposite direction, taking into account the extension speed factor "FFR". Then the tool is returned in rapid traverse to the load position.

Safety distance SD*

This parameter specifies where drilling machining starts. This position is approached in rapid traverse. The dimension is absolute.

Depth DT*

This parameter specifies where machining stops.

The dimension is absolute.

Thread pitch P*

Pitch of the thread.

Feed factor retraction FFR

Factor for retraction. Default value: FFR = 2

Diameter D

Diameter of the threaded hole. Standard "0".

Axis group AGR

Number of the axis group (default: default axis group).

Tool spindle TSP

Number of the tool spindle: 0 = S, 1 = S1; 2 = S2. The default is in "Options".

Tool direction of rotation TDR

Direction of rotation of the tool. Right=TDR1, left TDR-1. Standard in "Options".

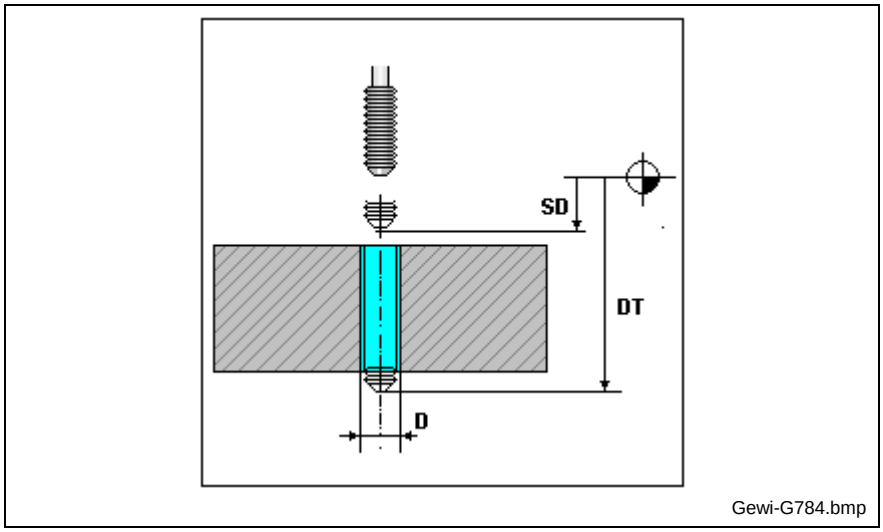


Fig. 2-28: G784 Rigid tapping

G785 Centric Rigid Tapping

Function Rigid tapping cycle G785 implements the entire movement sequence, including auxiliary movements, that is required for threading with a thread tapper. Contrary to cycle G784, this cycle includes interpolation between the main spindle and the feed axis. This means rigid tapping is possible; no mechanical floating head tap-holder is required.

Process:

In rapid traverse, the tool is positioned from the previously approached position to the center of rotation and then to safety distance "SD". The next step is to drill with feed to depth "DT" and then to reposition to safety distance "SD" in the feed with the spindle rotating in the opposite direction, taking into account the extension speed factor "FFR". Then the tool is returned in rapid traverse to the load position.

Safety distance SD* This parameter specifies where drilling machining starts. This position is approached in rapid traverse. The dimension is absolute.

Depth DT* This parameter specifies where machining stops.

The dimension is absolute.

Thread pitch P* Pitch of the thread.

Feed factor retraction FFR Factor for retraction. Default value: FFR = 2.

Diameter D Diameter of the threaded hole. Standard "0".

Axis group AGR Number of the axis group (default: default axis group).

Tool spindle TSP Number of the tool spindle: 0 = S, 1 = S1; 2 = S2. The default is in "Options".

Tool direction of rotation TDR Direction of rotation of the tool. Right=TDR1, left TDR-1. Standard in "Options".

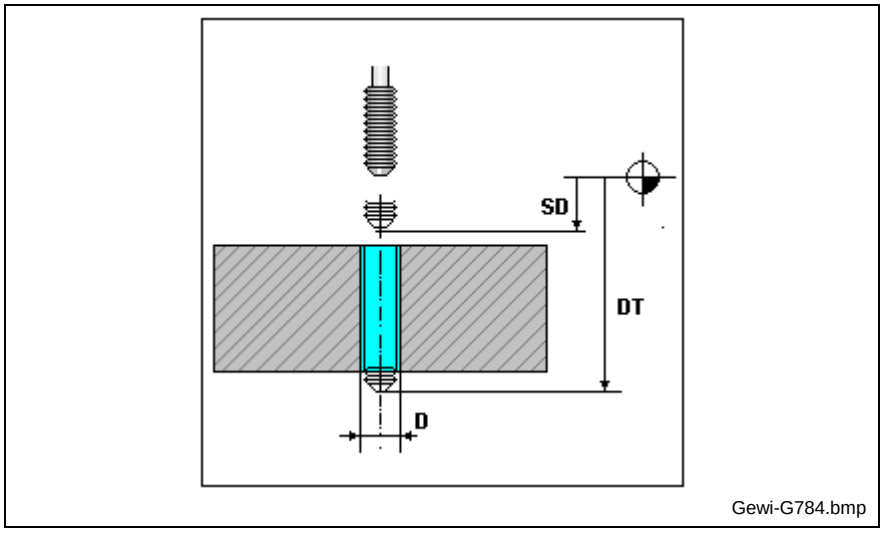


Fig. 2-29: G785 Rigid tapping

G776 Thread Cutting

Function

Thread cutting cycle G776 implements the entire movement sequence, including auxiliary movements, that is required for cutting a thread.

Process:

The cycle recognizes whether outside or inside machining is to be performed by means of the value of variables Starting point "SX" and End point "EX". This is followed by feed in rapid traverse in direction X for the amount of the first feed increment "NI" while taking flank angle "EA" in direction Z into account. This is followed by straight turning, synchronous to the applied speed of the main spindle with pitch "P" up to end point "EZ". Then, in rapid traverse in direction X, a 1mm lift above the thread occurs, followed by withdrawal to starting point Z "SZ". The feed, straight cutting, lifting and retraction operations are repeated until end point "EX". Feed is reduced in this case to keep the chip cross-section constant.

The reduced feed is calculated from the first feed increment "NI" multiplied by the square root of the i^{th} run. The number of finishing runs "GC" is then processed.

Moving to start position "SX" in rapid traverse ends the cycle.

Then movement to the tool retract position occurs.

Starting point SX* This parameter specifies the starting diameter of the thread.

Starting point SZ* This parameter specifies the starting length of the thread.

End point EX* This parameter specifies the final diameter of the thread.

End point EZ* This parameter specifies the final length of the thread.

Feed increment NI* First feed depth - as an increment.

Tool retraction TRV When cutting is complete, there are four tool retraction variants. They differ by the method that is used to determine whether and how the "Starting position" that was programmed **before** the cycle is to be attained at the end of the cycle. The default value can be specified under "Options":

- **TRV0** Tool does **not** travel to the starting position - it stops after the last step.
- **TRV1** Tool travels to the starting position simultaneously in both axes.
- **TRV2** Tool travels to the starting position first in the flat axis and then in the longitudinal axis.
- **TRV2** Tool travels to the starting position first in the longitudinal axis and then in the flat axis.

Cutting variant CV Specifies how the thread depth is to be fed. CV1 = radial symmetric feed. CV2 = flank feed. CV3 = alternating feed.

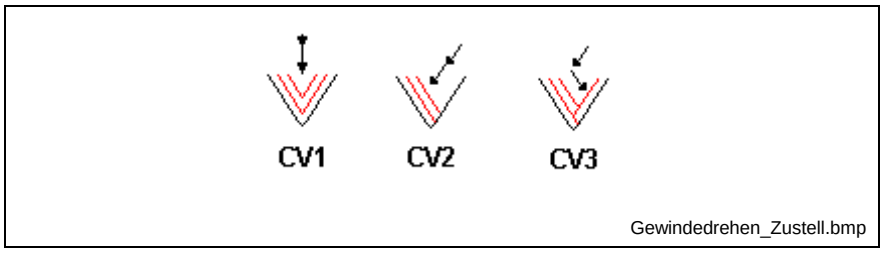


Fig. 2-30: Thread cutting variants

Thread pitch P Pitch of the thread.

Finishing runs GC Number of finishing runs (idle runs)

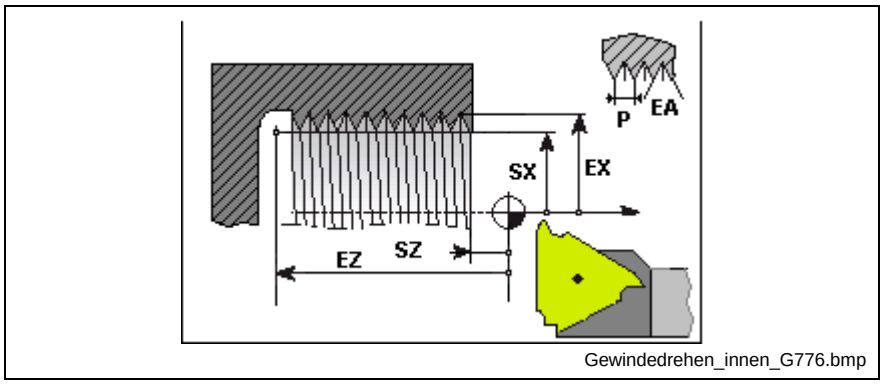


Fig. 2-31: Inside thread cutting

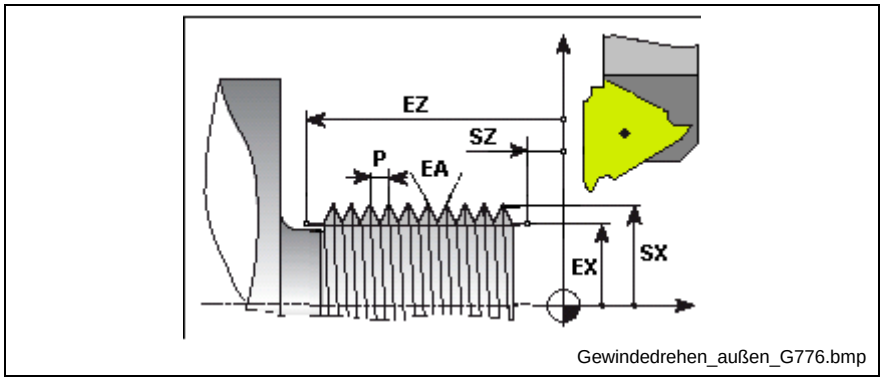


Fig. 2-32: Outside thread cutting

3 Machine Parameters and Configuration

3.1 General Statements

The compiler and the dialog boxes are highly dependent on the machine parameters of the processes for which the NC program is generated.

The relevant machine parameters are provided in an INI file outside of the interface for graphic NC programming and outside of the compiler. This makes it possible to relieve the load on the function interface if ID information (modification time) of the parameter block is checked to see whether the configuration file still is suitable for the current machine parameters. If there are differences, the machine data are recompiled. In addition, external development and testing are simplified.

The configuration file can also store user settings (e.g. possible axis groups) and states (e.g. screen limits).

A separate INI file is created for every process. This can be found in the CustomData path.

Name structure: GnpConfig_XXX_YYY.ini. XXX three-digit device number with leading zeroes, YYY three-digit process number with leading zeroes.

3.2 Relevant Machine Parameters

System Parameters

Number of Tool Edges

The number of tool edges is to be heeded during plunging and die cycles.

MTC 200 parameters

Tool management

A00.052 = 0 no

A00.052 = 1 yes

Maximum number of tool edges

Relevant only if Tool management has been answered with "yes" (A00.052). Otherwise, there are no tool edges.

A00.054 = 1 - 9

Parameters in configuration file

Section [PARAMETER]

Value address: MaxEdges

Values: 0...9

Axes and Axis Types

Axes and axis types can be found in the axis parameters.

Number of relevant NC axes

Parameters in configuration file

Section [PARAMETER]
Value address: MaxIndexAxes
Values: Number >= 0

Process Parameters

The following process parameters are to be taken into account:

Default system of units for programming

This datum can be used, for example, for internally generated default parameters such as lifting movements, etc.

Note: Note that all contour definitions and all generated NC blocks are expected/output in the default system of units.

MTC 200 parameters
Bxx.001 = 0 mm
Bxx.001 = 1 inches

Parameters in configuration file
Section [PARAMETER]
Value address: BasicUnit
Values: mm/inches

Programmable decimal places for distances

MTC 200 parameters
Bxx.002 = 4 or 5

Parameters in configuration file
Section [PARAMETER]
Value address: Precision
Values: 4 / 5

Default interpolation plane

MTC 200 parameters
Bxx.004 = 0 G17
Bxx.004 = 1 G18
Bxx.004 = 2 G19

Parameters in configuration file
Section [PARAMETER]
Value address: BasicPlane
Values: G17/G18/G19

Default for radius/diameter programming

MTC 200 parameters
Bxx.013 = 0 G15 (radius)
Bxx.013 = 1 G16 (diameter)

Parameters in configuration file
Section [PARAMETER]
Value address: BasicXScal
Values: G15/G16

Cartesian-polar coordinate transformation

MTC 200 parameters
Bxx.035 = 0 no
Bxx.035 = 1 yes

Parameters in configuration file
Section [PARAMETER]
Value address: Transformation
Values: 0/1

Axis Parameters

All axes are listed that are assigned to the process and which can obtain axis definition X, Y or Z.

Axis type (not in the axis parameters, otherwise in the system parameters.)

Parameters in configuration file
Section [AXIS_XXX] XXX = Index of the axis in the configuration file

Value address: Type
Values:

ANALOG_LINEAR_AXIS	analog linear axis
ANALOG_ROTARY_AXIS	analog rotary axis
ANALOG_MAIN_SPINDLE	analog main spindle
ANALOG_COMB_TURRET_AXIS	analog turret axis
C_AXIS	analog main spindle with rotary axis capability

DIGITAL_LINEAR_AXIS	digital linear axis
DIGITAL_ROTARY_AXIS	digital rotary axis
DIGITAL_MAIN_SPINDLE	digital main spindle
DIGITAL_COMB_TURRET_AXIS	digital turret axis
DIGITAL_C_AXIS	digital main spindle with rotary axis capability

Axis designation, axis designation 1, axis designation for main spindle mode

MTC 200 parameters Cxx.001 = X, Y, Z, U, V, W, A, B, C, S with and without index 1, 2, 3

Parameters in configuration file
Section file: [AXIS_XXX] XXX = Index of the axis in the configuration
Value address: Name
Values: X, Y, Z, U, V, W, A, B, C, S with and without index 1, 2, 3

Axis meaning (axis functions)

MTC 200 parameters Cxx.053 = X, Y, Z, U, V, W, A, B, C, S(S1), S2, S3

Parameters in configuration file
Section file: [AXIS_XXX] XXX = Index of the axis in the configuration
Value address: Function
Values: X, Y, Z, U, V, W, A, B, C, S(S1), S2, S3, separated by a comma (example: S1,Y)

Axis designation for rotary axis mode

Specified only for digital main spindles with rotary axis capability

MTC 200 parameters Cxx.054 = X, Y, Z, U, V, W, A, B, C with and without index 1, 2, 3

Parameters in configuration file
Section file: [AXIS_XXX] XXX = Index of the axis in the configuration
Value address: NameRA
Values: X, Y, Z, U, V, W, A, B, C, S with and without index 1, 2, 3

Axis number of allocated rotary axis

Specified only for digital main spindles

MTC 200 parameters Cxx.064 = Axis number (0-20)

Parameters in configuration file
Section file: [AXIS_XXX] XXX = Index of the axis in the configuration
Value address: AllocRA
Values: Index of the axis in the configuration file

Axis designation 2

MTC 200 parameters Cxx.075 = X, Y, Z, U, V, W, A, B, C with and without index 1, 2, 3

Parameters in configuration file
Section file: [AXIS_XXX] XXX = Index of the axis in the configuration
Value address: Name2
Values: X, Y, Z, U, V, W, A, B, C, S with and without index 1, 2, 3

Rotation direction for transformation

Specified for digital linear axes, digital rotary axes and digital main spindles with rotary axis capability.

If the reverse direction is switched on, the sense of rotation of the circle, the radius correction directions and the sign of the coordinates must be reversed in the postprocessor.

MTC 200 parameters
Cxx.076 = 0 no reversal
Cxx:077 = 1 direction reversal

Parameters in configuration file Section file: [AXIS_XXX] XXX = Index of the axis in the configuration

Value address: TransDir
Values: 0/1

3.3 Options and User Settings

ID Information of the Parameter Block

This is used for comparison with the active parameter block. If there are differences, the information from the parameter block is recompiled.

Section [PARAMID]

Value addresses and values:

Number = Number in the NC parameter directory [01-99]

Name = Name of the NC parameter record [max. 32 ASCII chars.]

Size = Length of the NC parameter record [bytes]

Date = Date of creation/last change of the parameter record [DD.MM.YY]

Time = Time of creation/last change of the parameter record [HH:MM:SS]

Device = Device number

Process = Process number

Axis Groups

These are the possible axis groups that are offered in the selection dialog boxes. If the default axis group is not present, it is inserted automatically.

Section: [AxisGroups]

- Number of axis groups
  Value address Max_Index_Data
  Values Number
- Index of the default axis group
  Value address IndexDefault
  Values Index (>=1)
- In front of / behind the center of rotation
  The view and the dialog box sequence (help pictures for contour movement definition) can be switched to "In front of the center of rotation" (positive X-axis downwards) for turning contours and turning cycles.
  Value address InFrontOfCentre
  Values 0 behind / 1 in front of center
- Axis group settings
  Description of the axis group with NC commands
  Value address Settings_XXX (XXX is the index of the axis group)
  Values Strings, e.g. G15 G31 X1 Y1 Z
  RI for G32 is missing here.
  See the description of the axis groups
- Comment for axis group (for selection in a combo box)
  Value address Comment_XXX (XXX is the index of the axis group)
  Values any string
- Default window
  Specified default window for the axis group if no geometry has yet been defined for this axis group.

Value address Window_XXX
Values left, bottom, right, top (coordinates)

Technical/Technological Data Presets

Technological data that modify machining are stored here.

Section: [Tech_Settings]

- Tool addressing
- Value address Tool_ADR
  Values 0 = using memory and location number; 1=using tool and duplo number
- Tool edge radius correction for turning
- Value address Edge_Korr (only "0" has been implemented)
  Values 0 = without; 1 = only for contour cut;
  2= roughing and contour cut
- Tool edge radius comparison (Delta)
- Value address Edge_Delta
  Values Permitted difference between the actual tool radius and the programmed one.
- Tangential safety distance / approach in [mm]
- Value address Siabst_Tan
  Values
- Tool retraction turning - outside machining
- Value address Tool_Rev_T_Out
  Values 0-3
- Tool retraction turning - inside machining
- Value address Tool_Rev_T_In
  Values 0-3
- Tool retraction plunging - radial machining
- Value address Tool_Rev_GR_TEP68
  Values 0-3
- Tool retraction plunging - axial machining
- Value address Tool_Rev_GR_TEP57
  Values 0-3
- Cutting variant Turning
- Value address Cut_Var
- Values 1-7
- Tool departure path at the end of cutting [mm]
- Value address Cut_Rev_Path
  Values >0
- Tool departure angle at the end of cutting [deg.]
- Value address Cut_Rev_Ang
  Values 0-90
- Angle reduction for chip removal cycle [deg.]
- Value address Cut_Dc_Ang
  Values 0-30
- Cutting depth for chip removal cycle
- Value address Cutting_depth
  Values >0
- Tool radius for turning

- Value address Tool_R
  Values >0
- Number of the default tool spindle for G784, G785
- Value address Tool_TSP1
  Values 0..3
- Default tool direction of rotation for G784, G785
- Value address Tool_TDR1
  Values 1 = right, -1 = left
- Number of the default tool spindle for G884, G885, G888, G889
- Value address Tool_TSP2
  Values 0..3
- Default tool direction of rotation for G884, G885, G888, G889
- Value address Tool_TDR2
  Values 1 = right, -1 = left
- Maximum length of flat linear patterns within monotonously ascending contour areas that are **not** to be subjected to the RSF allowance for flat surfaces.
- Value address Remaining_LVI
  Values >=0
- Maximum length of longitudinal linear patterns (cylinder surfaces) within monotonously ascending contour areas that are **not** to be subjected to the RSL allowance for longitudinal surfaces.
- Value address Remaining_LHI
  Values >=0
- Maximum length of flat linear patterns within rear cuts that are **not** to be subjected to the RSF allowance for flat surfaces.
- Value address Remaining_LVII
  Values >=0
- Maximum length of longitudinal linear patterns (cylinder surfaces) within undercuts that are **not** to be subjected to the RSL allowance for longitudinal surfaces.
- Value address Remaining_LHII
  Values >=0
- Thread cutting variant
- Value address Thread_Var
  Values >1,2,3

4 Index

O

P

R

S

T

U

W

5 Service & Support

5.1 Helpdesk

Unser Kundendienst-Helpdesk im Hauptwerk Lohr am Main steht Ihnen mit Rat und Tat zur Seite. Sie erreichen uns

Our service helpdesk at our headquarters in Lohr am Main, Germany can assist you in all kinds of inquiries. Contact us

- telefonisch - by phone: **49 (0) 9352 40 50 60**
  über Service Call Entry Center - via Service Call Entry Center
  Mo-Fr 07:00-18:00 Mo-Fr 7:00 am - 6:00 pm
- per Fax - by fax: **+49 (0) 9352 40 49 41**
- per e-Mail - by e-mail: **service.svc@boschrexroth.de**

5.2 Service-Hotline

Außerhalb der Helpdesk-Zeiten ist der Service direkt ansprechbar unter

After helpdesk hours, contact our service department directly at

+49 (0) 171 333 88 26

oder - or **+49 (0) 172 660 04 06**

5.3 Internet

Unter **www.boschrexroth.com** finden Sie ergänzende Hinweise zu Service, Reparatur und Training sowie die **aktuellen** Adressen *) unserer auf den folgenden Seiten aufgeführten Vertriebs- und Servicebüros.

Verkaufsniederlassungen
Niederlassungen mit Kundendienst

Außerhalb Deutschlands nehmen Sie bitte zuerst Kontakt mit unserem für Sie nächstgelegenen Ansprechpartner auf.

*) Die Angaben in der vorliegenden Dokumentation können seit Drucklegung überholt sein.

At **www.boschrexroth.com** you may find additional notes about service, repairs and training in the Internet, as well as the **actual** addresses *) of our sales- and service facilities figuring on the following pages.

sales agencies
offices providing service

Please contact our sales / service office in your area first.

*) Data in the present documentation may have become obsolete since printing.

5.4 Vor der Kontaktaufnahme... - Before contacting us...

Wir können Ihnen schnell und effizient helfen wenn Sie folgende Informationen bereithalten:

1. detaillierte Beschreibung der Störung und der Umstände.
2. Angaben auf dem Typenschild der betreffenden Produkte, insbesondere Typenschlüssel und Seriennummern.
3. Tel.-/Faxnummern und e-Mail-Adresse, unter denen Sie für Rückfragen zu erreichen sind.

For quick and efficient help, please have the following information ready:

1. Detailed description of the failure and circumstances.
2. Information on the type plate of the affected products, especially type codes and serial numbers.
3. Your phone/fax numbers and e-mail address, so we can contact you in case of questions.

5.5 Kundenbetreuungsstellen - Sales & Service Facilities

Deutschland – Germany

vom Ausland: (0) nach Landeskennziffer weglassen!
from abroad: don't dial (0) after country code!

Vertriebsgebiet Mitte Germany Centre	**SERVICE** **CALL ENTRY CENTER** **MO – FR** **von 07:00 - 18:00 Uhr** from 7 am – 6 pm	**SERVICE** **HOTLINE** **MO – FR** **von 17:00 - 07:00 Uhr** from 5 pm - 7 am **+ SA / SO**	**SERVICE** **ERSATZTEILE / SPARES** verlängerte Ansprechzeit - extended office time - ♦ nur an Werktagen - only on working days - ♦ von 07:00 - 18:00 Uhr - from 7 am - 6 pm -
Rexroth Indramat GmbH Bgm.-Dr.-Nebel-Str. 2 / Postf. 1357 97816 Lohr am Main / 97803 Lohr **Kompetenz-Zentrum Europa** Tel.: +49 (0)9352 40-0 Fax: +49 (0)9352 40-4885	**Tel. +49 (0) 9352 40 50 60** service.svc@boschrexroth.de	**Tel.: +49 (0)172 660 04 06** **oder / or** **Tel.: +49 (0)171 333 88 26**	**Tel. +49 (0) 9352 40 42 22**
Vertriebsgebiet Süd Germany South	Vertriebsgebiet West Germany West	Gebiet Südwest Germany South-West	
Bosch Rexroth AG Landshuter Allee 8-10 80637 München Tel.: +49 (0)89 127 14-0 Fax: +49 (0)89 127 14-490	Bosch Rexroth AG Regionalzentrum West Borsigstrasse 15 40880 Ratingen Tel.: +49 (0)2102 409-0 Fax: +49 (0)2102 409-406 +49 (0)2102 409-430	Bosch Rexroth AG Service-Regionalzentrum Süd-West Siemensstr.1 70736 Fellbach Tel.: +49 (0)711 51046–0 Fax: +49 (0)711 51046–248	
Vertriebsgebiet Nord Germany North	Vertriebsgebiet Mitte Germany Centre	Vertriebsgebiet Ost Germany East	Vertriebsgebiet Ost Germany East
Bosch Rexroth AG Walsroder Str. 93 30853 Langenhagen Tel.: +49 (0) 511 72 66 57-0 Service: +49 (0) 511 72 66 57-256 Fax: +49 (0) 511 72 66 57-93 Service: +49 (0) 511 72 66 57-783	Bosch Rexroth AG Regionalzentrum Mitte Waldecker Straße 13 64546 Mörfelden-Walldorf Tel.: +49 (0) 61 05 702-3 Fax: +49 (0) 61 05 702-444	Bosch Rexroth AG Beckerstraße 31 09120 Chemnitz Tel.: +49 (0)371 35 55-0 Fax: +49 (0)371 35 55-333	Bosch Rexroth AG Regionalzentrum Ost Walter-Köhn-Str. 4d 04356 Leipzig Tel.: +49 (0)341 25 61-0 Fax: +49 (0)341 25 61-111

Europa (West) - Europe (West)

vom Ausland: (0) nach Landeskennziffer weglassen, Italien: 0 nach Landeskennziffer mitwählen
from abroad: don't dial (0) after country code, Italy: dial 0 after country code

Austria - Österreich Bosch Rexroth GmbH Electric Drives & Controls Stachegasse 13 1120 Wien Tel.: +43 (0)1 985 25 40 Fax: +43 (0)1 985 25 40-93	Austria – Österreich Bosch Rexroth GmbH Electric Drives & Controls Industriepark 18 4061 Pasching Tel.: +43 (0)7221 605-0 Fax: +43 (0)7221 605-21	Belgium - Belgien Bosch Rexroth AG Electric Drives & Controls Industrielaan 8 1740 Ternat Tel.: +32 (0)2 5830719 - service: +32 (0)2 5830717 Fax: +32 (0)2 5830731 service@boschrexroth.be	Denmark - Dänemark BEC A/S Zinkvej 6 8900 Randers Tel.: +45 (0)87 11 90 60 Fax: +45 (0)87 11 90 61
Great Britain – Großbritannien Bosch Rexroth Ltd. Electric Drives & Controls Broadway Lane, South Cerney Cirencester, Glos GL7 5UH Tel.: +44 (0)1285 863000 Fax: +44 (0)1285 863030 sales@boschrexroth.co.uk service@boschrexroth.co.uk	Finland - Finnland Bosch Rexroth Oy Electric Drives & Controls Ansatie 6 017 40 Vantaa Tel.: +358 (0)9 84 91-11 Fax: +358 (0)9 84 91-13 60	France - Frankreich Bosch Rexroth SAS Electric Drives & Controls Avenue de la Trentaine (BP. 74) 77503 Chelles Cedex Tel.: +33 (0)164 72-70 00 Fax: +33 (0)164 72-63 00 **Hotline:** +33 (0)608 33 43 28	France - Frankreich Bosch Rexroth SAS Electric Drives & Controls ZI de Thibaud, 20 bd. Thibaud (BP. 1751) 31084 Toulouse Tel.: +33 (0)5 61 43 61 87 Fax: +33 (0)5 61 43 94 12
France – Frankreich Bosch Rexroth SAS Electric Drives & Controls 91, Bd. Irène Joliot-Curie 69634 Vénissieux – Cedex Tel.: +33 (0)4 78 78 53 65 Fax: +33 (0)4 78 78 53 62	Italy - Italien Bosch Rexroth S.p.A. Via G. Di Vittorio, 1 20063 Cernusco S/N.MI **Hotline:** +39 02 92 365 563 Tel.: +39 02 92 365 1 Service: +39 02 92 365 326 Fax: +39 02 92 365 500 Service: +39 02 92 365 503	Italy - Italien Bosch Rexroth S.p.A. Via Paolo Veronesi, 250 10148 Torino Tel.: +39 011 224 88 11 Fax: +39 011 224 88 30	Italy - Italien Bosch Rexroth S.p.A. Via Mascia, 1 80053 Castellamare di Stabia NA Tel.: +39 081 8 71 57 00 Fax: +39 081 8 71 68 85
Italy - Italien Bosch Rexroth S.p.A. Via del Progresso, 16 (Zona Ind.) 35020 Padova Tel.: +39 049 8 70 13 70 Fax: +39 049 8 70 13 77	Italy - Italien Bosch Rexroth S.p.A. Via Isonzo, 61 40033 Casalecchio di Reno (Bo) Tel.: +39 051 29 86 430 Fax: +39 051 29 86 490	Netherlands - Niederlande/Holland Bosch Rexroth Services B.V. Technical Services Kruisbroeksestraat 1 (P.O. Box 32) 5281 RV Boxtel Tel.: +31 (0) 411 65 16 40 +31 (0) 411 65 17 27 Fax: +31 (0) 411 67 78 14 +31 (0) 411 68 28 60 services@boschrexroth.nl	Netherlands – Niederlande/Holland Bosch Rexroth B.V. Kruisbroeksestraat 1 (P.O. Box 32) 5281 RV Boxtel Tel.: +31 (0) 411 65 19 51 Fax: +31 (0) 411 65 14 83 www.boschrexroth.nl
Norway - Norwegen Bosch Rexroth AS Electric Drives & Controls Berghagan 1 or: Box 3007 1405 Ski-Langhus 1402 Ski Tel.: +47 (0)64 86 41 00 Fax: +47 (0)64 86 90 62 **Hotline:** +47 (0)64 86 94 82 jul.ruud@rexroth.no	Spain - Spanien Bosch Rexroth S.A. Electric Drives & Controls Centro Industrial Santiga Obradors s/n 08130 Santa Perpetua de Mogoda Barcelona Tel.: +34 9 37 47 94 00 Fax: +34 9 37 47 94 01	Spain – Spanien Goimendi S.A. Electric Drives & Controls Parque Empresarial Zuatzu C/ Francisco Grandmontagne no.2 20018 San Sebastian Tel.: +34 9 43 31 84 21 - service: +34 9 43 31 84 56 Fax: +34 9 43 31 84 27 - service: +34 9 43 31 84 60 sat.indramat@goimendi.es	Sweden - Schweden Bosch Rexroth AB Electric Drives & Controls - Varuvägen 7 (Service: Konsumentvägen 4, Älfsjö) 125 81 Stockholm Tel.: +46 (0)8 727 92 00 Fax: +46 (0)8 647 32 77
Sweden - Schweden Bosch Rexroth AB Electric Drives & Controls Ekvändan 7 254 67 Helsingborg Tel.: +46 (0) 42 38 88 -50 Fax: +46 (0) 42 38 88 -74	Switzerland West - Schweiz West Bosch Rexroth Suisse SA Electric Drives & Controls Rue du village 1 1020 Renens Tel.: +41 (0)21 632 84 20 Fax: +41 (0)21 632 84 21	Switzerland East - Schweiz Ost Bosch Rexroth Schweiz AG Electric Drives & Controls Hemrietstrasse 2 8863 Buttikon Tel. +41 (0) 55 46 46 111 Fax +41 (0) 55 46 46 222	

Europa (Ost) - Europe (East)

vom Ausland: (0) nach Landeskennziffer weglassen
from abroad: don't dial (0) after country code

Czech Republic - Tschechien Bosch -Rexroth, spol.s.r.o. Hviezdoslavova 5 627 00 Brno Tel.: +420 (0)5 48 126 358 Fax: +420 (0)5 48 126 112	Czech Republic - Tschechien DEL a.s. Strojírenská 38 591 01 Zdar nad Sázavou Tel.: +420 566 64 3144 Fax: +420 566 62 1657	Hungary - Ungarn Bosch Rexroth Kft. Angol utca 34 1149 Budapest Tel.: +36 (1) 422 3200 Fax: +36 (1) 422 3201	Poland – Polen Bosch Rexroth Sp.zo.o. ul. Staszica 1 05-800 Pruszków Tel.: +48 22 738 18 00 – service: +48 22 738 18 46 Fax: +48 22 758 87 35 – service: +48 22 738 18 42
Poland – Polen Bosch Rexroth Sp.zo.o. Biuro Poznan ul. Dabrowskiego 81/85 60-529 Poznan Tel.: +48 061 847 64 62 /-63 Fax: +48 061 847 64 02	Romania - Rumänien East Electric S.R.L. Bdul Basarabia no.250, sector 3 73429 Bucuresti Tel./Fax:: +40 (0)21 255 35 07 +40 (0)21 255 77 13 Fax: +40 (0)21 725 61 21 eastel@rdsnet.ro	Romania - Rumänien Bosch Rexroth Sp.zo.o. Str. Drobety nr. 4-10, app. 14 70258 Bucuresti, Sector 2 Tel.: +40 (0)1 210 48 25 +40 (0)1 210 29 50 Fax: +40 (0)1 210 29 52	Russia - Russland Bosch Rexroth OOO Wjatskaja ul. 27/15 127015 Moskau Tel.: +7-095-785 74 78 +7-095 785 74 79 Fax: +7 095 785 74 77 laura.kanina@boschrexroth.ru
Russia - Russland ELMIS 10, Internationalnaya 246640 Gomel, Belarus Tel.: +375/ 232 53 42 70 +375/ 232 53 21 69 Fax: +375/ 232 53 37 69 elmis_ltd@yahoo.com	Turkey - Türkei Bosch Rexroth Otomasyon San & Tic. A..S. Fevzi Cakmak Cad No. 3 34295 Sefaköy - Istanbul Tel.: +90 212 413 34-00 Fax: +90 212 413 34-17	Slowenia - Slowenien DOMEL Otoki 21 64 228 Zelezniki Tel.: +386 5 5117 152 Fax: +386 5 5117 225 brane.ozebek@domel.si	

Africa, Asia, Australia – incl. Pacific Rim

Australia - Australien AIMS - Australian Industrial Machinery Services Pty. Ltd. 28 Westside Drive Laverton North Vic 3026 Melbourne Tel.: +61 3 93 14 3321 Fax: +61 3 93 14 3329 **Hotlines:** +61 3 93 14 3321 +61 4 19 369 195 enquires@aimservices.com.au	Australia - Australien Bosch Rexroth Pty. Ltd. No. 7, Endeavour Way Braeside Victoria, 31 95 Melbourne Tel.: +61 3 95 80 39 33 Fax: +61 3 95 80 17 33 mel@rexroth.com.au	China Shanghai Bosch Rexroth Hydraulics & Automation Ltd. Waigaoqiao, Free Trade Zone No.122, Fu Te Dong Yi Road Shanghai 200131 - P.R.China Tel.: +86 21 58 66 30 30 Fax: +86 21 58 66 55 23 richard.yang_sh@boschrexroth.com.cn _gf.zhu_sh@boschrexroth.com.cn	China Shanghai Bosch Rexroth Hydraulics & Automation Ltd. 4/f, Marine Tower No.1, Pudong Avenue Shanghai 200120 - P.R.China Tel: +86 21 68 86 15 88 Fax: +86 21 58 40 65 77
China Bosch Rexroth China Ltd. 15/F China World Trade Center 1, Jianguomenwai Avenue Beijing 100004, P.R.China Tel.: +86 10 65 05 03 80 Fax: +86 10 65 05 03 79	China Bosch Rexroth China Ltd. Guangzhou Repres. Office Room 1014-1016, Metro Plaza, Tian He District, 183 Tian He Bei Rd Guangzhou 510075, P.R.China Tel.: +86 20 8755-0030 +86 20 8755-0011 Fax: +86 20 8755-2387	China Bosch Rexroth (China) Ltd. A-5F., 123 Lian Shan Street Sha He Kou District Dalian 116 023, P.R.China Tel.: +86 411 46 78 930 Fax: +86 411 46 78 932	China Melchers GmbH BRC-SE, Tightening & Press-fit 13 Floor Est Ocean Centre No.588 Yanan Rd. East 65 Yanan Rd. West Shanghai 200001 Tel.: +86 21 6352 8848 Fax: +86 21 6351 3138
Hongkong Bosch Rexroth (China) Ltd. 6th Floor, Yeung Yiu Chung No.6 Ind Bldg. 19 Cheung Shun Street Cheung Sha Wan, Kowloon, Hongkong Tel.: +852 22 62 51 00 Fax: +852 27 41 33 44 alexis.siu@boschrexroth.com.hk	India - Indien Bosch Rexroth (India) Ltd. Electric Drives & Controls Plot. No.96, Phase III Peenya Industrial Area Bangalore – 560058 Tel.: +91 80 51 17 0-211...-218 Fax: +91 80 83 94 345 +91 80 83 97 374 mohanvelu.t@boschrexroth.co.in	India - Indien Bosch Rexroth (India) Ltd. Electric Drives & Controls Advance House, II Floor Ark Industrial Compound Narol Naka, Makwana Road Andheri (East), Mumbai - 400 059 Tel.: +91 22 28 56 32 90 +91 22 28 56 33 18 Fax: +91 22 28 56 32 93 singh.op@boschrexroth.co.in	India - Indien Bosch Rexroth (India) Ltd. S-10, Green Park Extension New Delhi – 110016 Tel.: +91 11 26 56 65 25 +91 11 26 56 65 27 Fax: +91 11 26 56 68 87 koul.rp@boschrexroth.co.in
Indonesia - Indonesien PT. Bosch Rexroth Building # 202, Cilandak Commercial Estate Jl. Cilandak KKO, Jakarta 12560 Tel.: +62 21 7891169 (5 lines) Fax: +62 21 7891170 - 71 rudy.karimun@boschrexroth.co.id	Japan Bosch Rexroth Automation Corp. Service Center Japan Yutakagaoka 1810, Meito-ku, NAGOYA 465-0035, Japan Tel.: +81 52 777 88 41 +81 52 777 88 53 +81 52 777 88 79 Fax: +81 52 777 89 01	Japan Bosch Rexroth Automation Corp. Electric Drives & Controls 2F, I.R. Building Nakamachidai 4-26-44, Tsuzuki-ku YOKOHAMA 224-0041, Japan Tel.: +81 45 942 72 10 Fax: +81 45 942 03 41	Korea Bosch Rexroth-Korea Ltd. Electric Drives and Controls Bongwoo Bldg. 7FL, 31-7, 1Ga Jangchoong-dong, Jung-gu Seoul, 100-391 Tel.: +82 234 061 813 Fax: +82 222 641 295
Korea Bosch Rexroth-Korea Ltd. 1515-14 Dadae-Dong, Saha-gu Electric Drives & Controls Pusan Metropolitan City, 604-050 Tel.: +82 51 26 00 741 Fax: +82 51 26 00 747 eunkyong.kim@boschrexroth.co.kr	Malaysia Bosch Rexroth Sdn.Bhd. 11, Jalan U8/82, Seksyen U8 40150 Shah Alam Selangor, Malaysia Tel.: +60 3 78 44 80 00 Fax: +60 3 78 45 48 00 hockhwa@hotmail.com rexroth1@tm.net.my	Singapore - Singapur Bosch Rexroth Pte Ltd 15D Tuas Road Singapore 638520 Tel.: +65 68 61 87 33 Fax: +65 68 61 18 25 sanjay.nemade @boschrexroth.com.sg	South Africa - Südafrika TECTRA Automation (Pty) Ltd. 71 Watt Street, Meadowdale Edenvale 1609 Tel.: +27 11 971 94 00 Fax: +27 11 971 94 40 **Hotline:** +27 82 903 29 23 georgv@tectra.co.za
Taiwan Bosch Rexroth Co., Ltd. Taichung Branch 1F., No. 29, Fu-Ann 5th Street, Xi-Tun Area, Taichung City Taiwan, R.O.C. Tel : +886 - 4 -23580400 Fax: +886 - 4 -23580402 charlie.chen@boschrexroth.com.tw jim.lin@boschrexroth.com.tw david.lai@boschrexroth.com.tw	Thailand NC Advance Technology Co. Ltd. 59/76 Moo 9 Ramintra road 34 Tharang, Bangkhen, Bangkok 10230 Tel.: +66 2 943 70 62 +66 2 943 71 21 Fax: +66 2 509 23 62 Hotline +66 1 984 61 52 sonkawin@hotmail.com		

Nordamerika – North America

USA Headquarters - Hauptniederlassung Bosch Rexroth Corporation Electric Drives & Controls 5150 Prairie Stone Parkway Hoffman Estates, IL 60192-3707 Tel.: +1 847 6 45 36 00 Fax: +1 847 6 45 62 01 servicebrc@boschrexroth-us.com repairbrc@boschrexroth-us.com	USA Central Region - Mitte Bosch Rexroth Corporation Electric Drives & Controls Central Region Technical Center 1701 Harmon Road Auburn Hills, MI 48326 Tel.: +1 248 3 93 33 30 Fax: +1 248 3 93 29 06	USA Southeast Region - Südwest Bosch Rexroth Corporation Electric Drives & Controls Southeastern Technical Center 3625 Swiftwater Park Drive Suwanee, Georgia 30124 Tel.: +1 770 9 32 32 00 Fax: +1 770 9 32 19 03	USA SERVICE-HOTLINE - 7 days x 24hrs - **+1-800-REX-ROTH** **+1 800 739 7684**
USA East Region – Ost Bosch Rexroth Corporation Electric Drives & Controls Charlotte Regional Sales Office 14001 South Lakes Drive Charlotte, North Carolina 28273 Tel.: +1 704 5 83 97 62 +1 704 5 83 14 86	USA Northeast Region – Nordost Bosch Rexroth Corporation Electric Drives & Controls Northeastern Technical Center 99 Rainbow Road East Granby, Connecticut 06026 Tel.: +1 860 8 44 83 77 Fax: +1 860 8 44 85 95	USA West Region – West Bosch Rexroth Corporation 7901 Stoneridge Drive, Suite 220 Pleasant Hill, California 94588 Tel.: +1 925 227 10 84 Fax: +1 925 227 10 81	
Canada East - Kanada Ost Bosch Rexroth Canada Corporation Burlington Division 3426 Mainway Drive Burlington, Ontario Canada L7M 1A8 Tel.: +1 905 335 5511 Fax: +1 905 335 4184 **Hotline:** +1 905 335 5511 michael.moro@boschrexroth.ca	Canada West - Kanada West Bosch Rexroth Canada Corporation 5345 Goring St. Burnaby, British Columbia Canada V7J 1R1 Tel. +1 604 205 5777 Fax +1 604 205 6944 **Hotline:** +1 604 205 5777 david.gunby@boschrexroth.ca	Mexico Bosch Rexroth Mexico S.A. de C.V. Calle Neptuno 72 Unidad Ind. Vallejo 07700 Mexico, D.F. Tel.: +52 55 57 54 17 11 Fax: +52 55 57 54 50 73 mariofelipe.hernandez@boschrexroth.com.mx	Mexico Bosch Rexroth S.A. de C.V. Calle Argentina No 3913 Fracc. las Torres 64930 Monterrey, N.L. Tel.: +52 81 83 65 22 53 +52 81 83 65 89 11 +52 81 83 49 80 91 Fax: +52 81 83 65 52 80 mario.quiroga@boschrexroth.com.mx

Südamerika – South America

Argentina - Argentinien Bosch Rexroth S.A.I.C. "The Drive & Control Company" Rosario 2302 B1606DLD Carapachay Provincia de Buenos Aires Tel.: +54 11 4756 01 40 +54 11 4756 02 40 +54 11 4756 03 40 +54 11 4756 04 40 Fax: +54 11 4756 01 36 +54 11 4721 91 53 victor.jabif@boschrexroth.com.ar	Argentina - Argentinien NAKASE Servicio Tecnico CNC Calle 49, No. 5764/66 B1653AOX Villa Balester Provincia de Buenos Aires Tel.: +54 11 4768 36 43 Fax: +54 11 4768 24 13 **Hotline:** +54 11 155 307 6781 nakase@usa.net nakase@nakase.com gerencia@nakase.com (Service)	Brazil - Brasilien Bosch Rexroth Ltda. Av. Tégula, 888 Ponte Alta, Atibaia SP CEP 12942-440 Tel.: +55 11 4414 56 92 +55 11 4414 56 84 Fax sales: +55 11 4414 57 07 Fax serv.: +55 11 4414 56 86 alexandre.wittwer@rexroth.com.br	Brazil - Brasilien Bosch Rexroth Ltda. R. Dr.Humberto Pinheiro Vieira, 100 Distrito Industrial [Caixa Postal 1273] 89220-390 Joinville - SC Tel./Fax: +55 47 473 58 33 Mobil: +55 47 9974 6645 prochnow@zaz.com.br
Columbia - Kolumbien Reflutec de Colombia Ltda. Calle 37 No. 22-31 Santafé de Bogotá, D.C. Colombia Tel.: +57 1 368 82 67 +57 1 368 02 59 Fax: +57 1 268 97 37 reflutec@neutel.com.co reflutec@007mundo.com			

Notes

Rexroth
Bosch Group

Bosch Rexroth AG
Electric Drives and Controls
P.O. Box 13 57
97803 Lohr, Germany
Bgm.-Dr.-Nebel-Str. 2
97816 Lohr, Germany
Phone +49 93 52-40-50 60
Fax +49 93 52-40-49 41
service.svc@boschrexroth.de
www.boschrexroth.com

Printed in Germany
DOK-MTC200-GNP*****V23-AW01-EN-P