Rexroth
Bosch Group

Rexroth MTC 200 NC Simulation

R911298710
Edition 01

Application Manual

Title

Rexroth MTC 200
NC Simulation

Type of Documentation Application Manual

Document Typecode DOK-MTC200-NC**SIM*V23-AW01-EN-P

Internal File Reference Document Number 120-1701-B307-01/EN

Purpose of Documentation This documentation describes the utilization of NC simulation for lathes.

Record of Revisions

Description	Release Date	Notes
120-1701-B307-01/EN	01.2004	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 Notes on Simulation

1.1 Preface

Function Simulation is an efficient graphic aid for visually checking parts programs, with an emphasis on geometric aspects, such as the dimensional accuracy of the workpiece, possible collisions, the contact conditions of the workpiece, etc. This is what is called a workpiece simulation.

Operational method The simulation presented herein is based on the interpretation of the parts program outside of the CNC and is **to a great extent independent** of its operating status. The movement instructions are converted into an animation which shows the workpiece in its particular machining state, the tool edge and, if necessary, line graphics corresponding to the programmed movement instructions.

Ordering information The simulation is an optional component of the user interface for the Rexroth MTC 200 and can be ordered under the type code SWS-MTC200-CVT-23VRS-MS.

1.2 Requirements and Conditions for Use

License Being an optional component, the simulation requires a license. The license can be ordered under Main Menu/Start Setup/License Management/Code.

Parameter set The simulation can only be used if the “Tool technology” system parameter is set to “turning/milling”. It is available only for those processes with which at least two linear main axes (first assigned process in the NC axis definition) and a tool management function are associated.

Machine tool The simulation can only and exclusively be used for turning machines. It can be used for the simulation of mere turning but also drilling and milling operations by driven tools, where either the physical NC axes or fictitious axes (end face operations with polar coordinate transformation or enclosure operations by cylinder coordinate transformation) can be optionally addressed.

NC program The NC simulation is based on a partial simulation of the CNC. As a consequence, the syntax that can be used in an NC program accepted syntactically by the interpreter of the simulator is restricted. Almost all instructions that are, in general, used in workpiece programs are implemented in the interpreter of the simulator. The instructions that are not considered in the simulator can rather be found in special subroutines (e.g. tool change) and in main programs that are executing managing and control functions. The restrictions on the part of the NC program are described in more detail below.

1.3 Simulation Input Data

Parts Program

The only input data used for the simulator is the NC program selected for simulation by the user. Further NC data, such as zero offset lists, NC variables, etc., that are used in an NC program run at the CNC, are not taken into consideration for the simulation. For that reason, there are restrictions to the quality of the simulation (e.g. accounting for zero shifts) or to the use of start requirements that are defined outside of the NC program (e.g. conditional jumps in relation to NC variables that have not been initialized in the NC program beforehand).

On the other hand, this approach permits the programmer to test his NC program practically in each stage of the programming process without having prepared all NC data, such as tool lists and zero offset lists, beforehand.

Since only one NC program can be simulated, jumps to other NC programs (JMP command) are not permitted.

Additional information in the parts program

The programmer may use the control program to provide additional information for the simulation, that is suitable to improve the graphical quality of the simulation. However, this is not mandatory.

Such additional information can, for instance, be used to describe

- the stock and
- the tool shapes.

To achieve this, a special syntax is used in the NC comments (parentheses) (refer to Chapter 2 "Additional Simulation Information in the Parts Program").

Note: The length of NC comments permitted in the CNC is limited to 80 characters. On downloading the program, longer NC comments are cut off, but do not cause an error. To call the programmer's attention to the limited length, any comment that is too long is represented in red in the NC program editor. Since, however, the length is not limited for the simulation, the red color can be ignored.

Cycle Package and Tool List

Preparatory steps for simulation

Among others, the following steps are implemented while the NC program is prepared for simulation:

- All user subroutines used are added to the parts program to be transferred to the simulation, and
- tool information is added, unless this has already been done by the programmer.

In the preparation for simulation, the currently loaded cycle package and the loaded tool list are used for deriving tool descriptions from the tool dimensions and to obtain the content of cycles (also refer to Chapter 3.1 "Preparatory Steps").

Note: By evaluating the tool list and cycle package loaded to the CNC, the preparation for simulation is the only issue where the simulation is dependent on the states of the CNC.

1.4 Syntactic and Other Restrictions

Syntactic restrictions

There are several instructions that are not permitted. Most of these instructions are listed below (incomplete list):

- Jumps to other programs (JMP, JSR)
- Setting and resetting of events as well as conditional jumps that are using events
- Access to NC and other data (OTD, TLD, AXD, DCD, MTD)
- Asynchronous treatment of events (process synchronization and event monitoring)
- Axis transfer among processes (FAX, GAX)
- Commands for tool management, motion commands for tool storages, and test functions associated with tool management, etc. (e.g. TSM, TMS, BTE, MOP, TCH) Preferrably, a subroutine (e.g. “M6”) should be used for tool change. It is also possible to write MMP and MTP.

Note: The simulation always signals a **syntax error,** irrespective of whether the syntax used in an NC block is really wrong or whether the simulation itself is not able to process a command.

Program structure

The simulation also requires that the NC program meets the following requirements (in addition to others):

- A variable may only be used in expressions or conditions, when it has been assigned in the NC program beforehand.
- Conditions that are determined outside of the NC program are not permissible.
- The NC program must not comprise any endless loops.

Remedies for programs not suitable for simulation

If a parts program is not suitable for simulation for syntactic or other reasons, there are two remedies:

- Copy the NC program and simplify or remove the instructions concerned.
- Exclude critical NC blocks from the simulation by NC instructions with a special syntax.

Note: It is not possible to simulate NC programs that are no parts programs, i.e. do not contain any motion instructions.

Graphical restrictions

In addition to commands that cannot be processed syntactically by the simulation, there are other commands which will not become effective in graphical regard although they can be written.

- Conversion of the length measurement unit G70/G71
- Rotational angle P in case of coordinate shifts G50, G51, G52
- All commands evaluating information stored in NC data: D-command, O-command, G54…G59

1.5 Control Panel

Call

The control panel is the control element for the virtual NC. It is a separate application that resides in the “bin” subdirectory of the installation as VirtualControlPanel.exe. To start the application, a link can be created on the desktop or in the start menu.

It is also possible to call the VNC panel via an F-key that has been configured by the user himself (refer to WinHMI Graphical User Interface – Configuration of Other Components – Projecting OP-Keys, F-Keys and M-Keys).

Boundary conditions

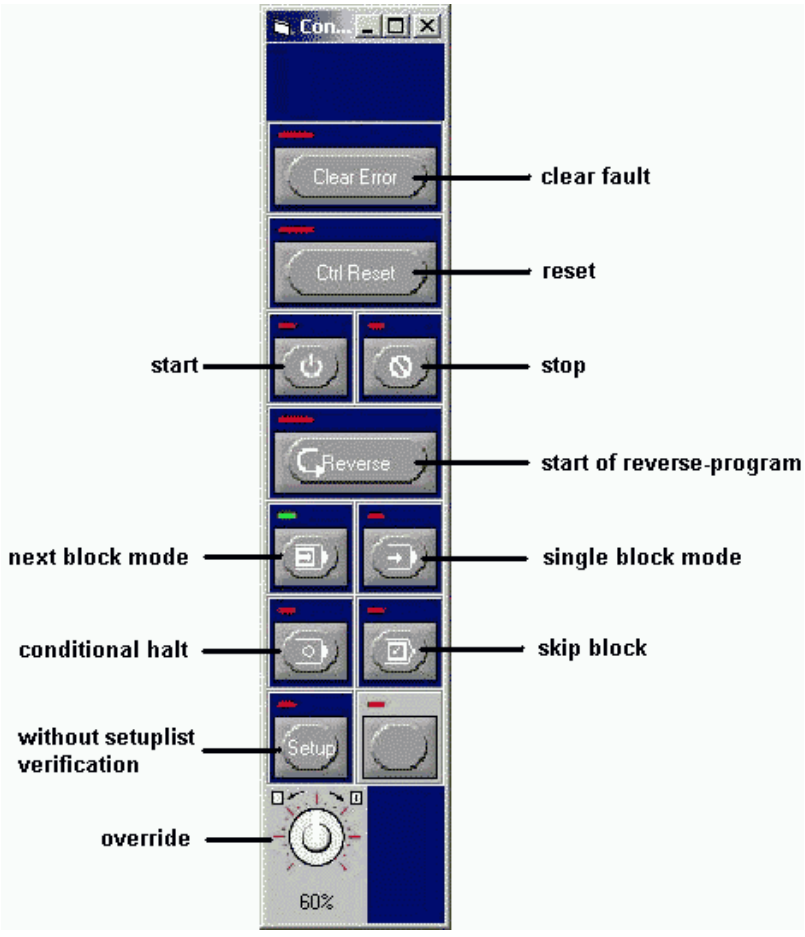
The control panel manipulates the control that is activated on the interface and the active process, if a virtual NC is selected. The control keys are hidden if a virtual control has not been selected.

Handling

If they are hidden, the control keys can be shown by pressing the <ALT>+<S> hot keys. Pressing these hot keys again will hide the control keys.

Operating the keys

The following figure shows the keys available on the control panel and their names.



VirtualControlPanel.bmp

Fig. 1-1: Control panel

2 Additional Simulation Information in the Parts Program

2.1 Stock Description

General Notes

Objective

The stock description enables the simulation to create a three-dimensional model of the workpiece and to simulate the cutting process as a removal of volume. The stock description is used for screen dimensioning. The respective machining state is used for collision monitoring. The 3D model is also necessary for calculating sectional views (full section, half section, any other section).

Preferrably, the stock description should be added after program start and before starting the motion instructions.

Missing stock description

Without the stock description, it is only possible to simulate the traversing motions, not the cutting process. In this case, screen dimensioning is based on the distances covered by feed motions.

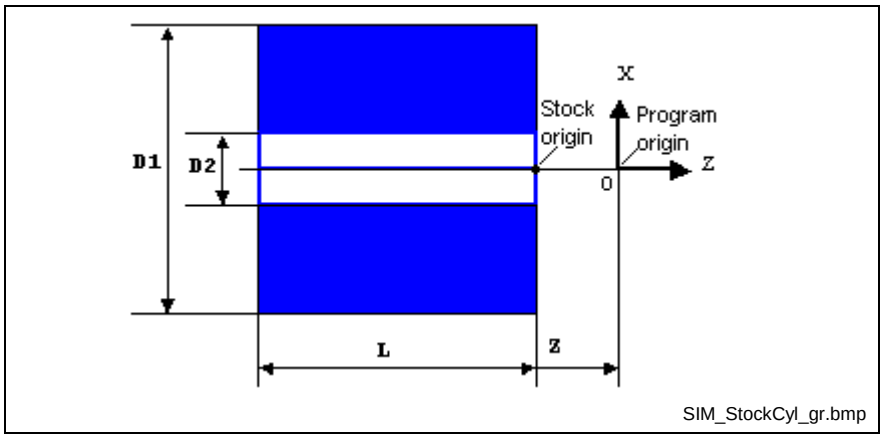
Multiple stock part description

Only the first of several stock descriptions contained in a parts program will become effective.

Cylindrical Stock

Intended use

Stocks having the shape of a cylinder are intended for rod sections which may, optionally, also comprise an internal through hole.



L: L Length
Z Zero shift to front end
D1 Outside diameter
D2 Inside diameter

Fig. 2-1: Cylindrical stock with hole

Syntax `(STOCK/L,D1,D2,Z)`

If a stock without internal hole is concerned, D2 should be zero.

Example (STOCK/125,80,30,-5)

Preformed Stock (Stock Contour)

Intended use

Usually, defining a preformed stock is reasonable for cast, forged or pre-turned workpieces.

Syntax

The stock contour is described by motion blocks, using the known syntax of the Rexroth MTC 200 for G1, G2 and G3. Each corner point of the contour is described by its own record block and in parentheses. It must be ensured that the contour is closed. If this is not the case, the last contour point is internally connected to the starting point via a straight line.

```
(STOCK/BEGIN)
(G1 Z{} X{}); starting point
(..); one/several contour blocks with G1, G2 or G3
(STOCK/END)
```

Example

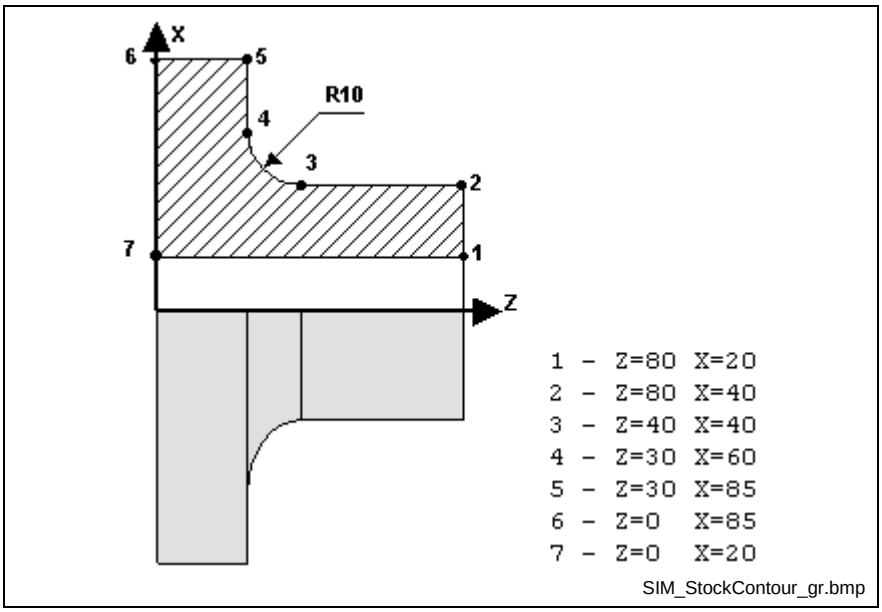


Fig. 2-2: Stock contour example

(STOCK/BEGIN)
(G1 Z80 X20)
(Z80 X40)
(Z40 X40)
(G2 Z30 X60 R10)
(G1 Z30 X85)
(Z0 X85)
(Z0 X20)
(Z80 X20)
(STOCK/END)

2.2 Starting Point

Intended use

A specified starting point is intended to take a starting position of the NC axes, that is “coincidental” in case of a real NC program start and will often be a tool changing position, into consideration in the simulation. It is especially recommended to define a starting point whenever the movement to the first machining position is to be presented graphically and coordinates are not specified from the first approach block in all axis directions.

Note: It is optional to define the starting point. If defined, it must be written **before** the first motion block.

Starting point definition is not realized in the present version.

Syntax

```
(FROM/Z,X)
```

Example

(FROM/200,400)

G0 X0

Z5; safety position for drilling

2.3 Tool Description

General Notes

Objective

The tool description only describes the tool edge part. In combination with the movement, the shape of the tool edge part defines the volume removed and, thus, the shape of the workpiece. The description of the tool edge determines a further requirement for workpiece modelling and also for collision considerations which are, however, only restricted to the tool edge part described (collision of the tool edge with the workpiece in rapid traverse blocks). The definition of the tool edge radius or tool diameter is, in addition, included in the tool radius compensation as equidistance.

Note: As an option, tool descriptions may also be added manually. However, missing tool descriptions may also be supplemented automatically while simulation is being prepared.

Position in the NC program

The tool information should be added in the NC program immediately after the tool change. If, via tool edge change (E-command), a different tool shape (double tools) is selected or the faces of recessing tools are changed while using more than two tool edges E, the tool description must be repeated.

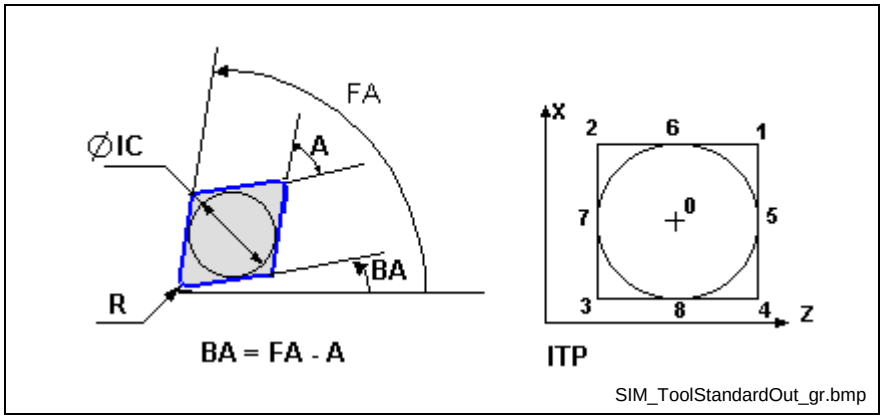
Turning Tools

Single-Edged Turning Tools (Contour Cutting Tool)

Intended use

The contour cutting tool description is used for single-edged tools with rhombic or triangular disposable insert, but also for special tools. Here, the basic shape of the tool edge is always based on a rhombic insert.

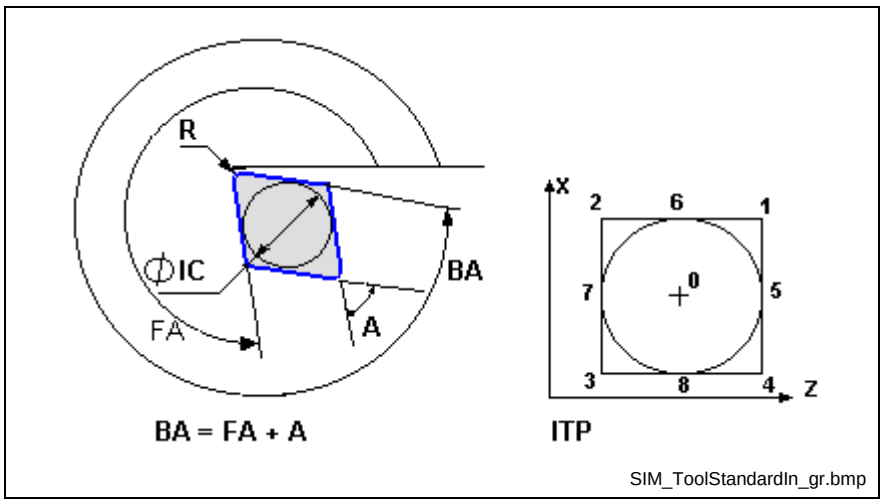
Syntax



L:		
	A	Insert angle
	BA, FA	Tool edge angle
	R	Tool edge radius
	IC	Insert size
	ITP	Tool edge orientation code

Fig. 2-3: Contour turning tool (outside)

Inside tools are dimensioned analogously to outside tools.



L:		
	A	Insert angle
	BA, FA	Tool edge angle
	R	Tool edge radius
	IC	Insert size
	ITP	Tool edge orientation code

Fig. 2-4: Contour turning tool (inside)

```
(TOOL/STANDARD,BA,A,R,IC,ITP)
```

Examples

Left-hand radial outside turning tool

(TOOL/STANDARD,10,75,0.8,10,3)

Left-hand axial inside turning tool

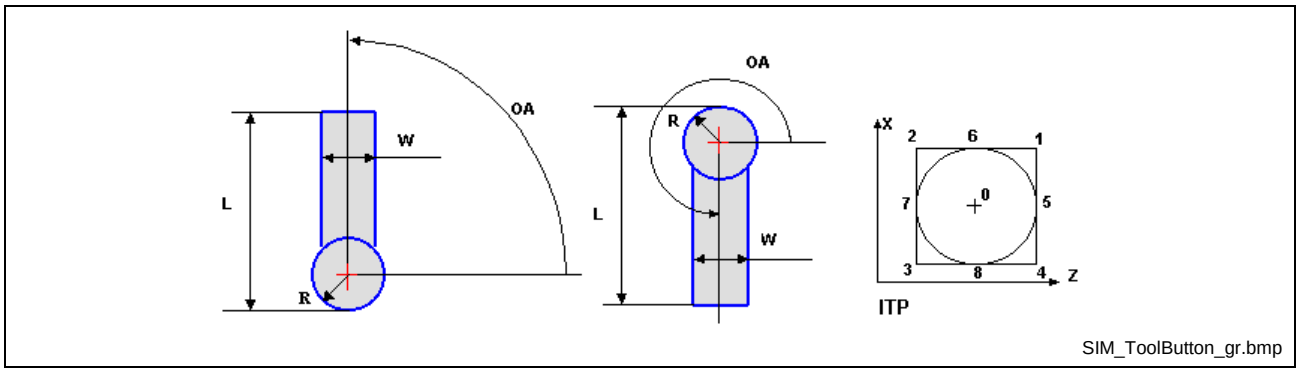
(TOOL/STANDARD,350,75,0.8,10,2)

Circular Tool

Intended use

Below follows a description of tools with circular insert. These are always **single-edged** tools. If a circular tool has two edges, then the necking tool description must be used.

Syntax



R Tool edge radius
L Length
W Holder width
OA Orientation angle
ITP Tool edge orientation code

Fig. 2-5: Circular tool (outside and inside)

```
(TOOL/BUTTON,R,L,W,OA,ITP)
```

Examples

Outside tool

(TOOL/BUTTON,6,30,10,90,8)

Inside tool

(TOOL/BUTTON,6,30,10,270,6)

Double-Edged Turning Tools (Necking Tool)

Intended use

The necking tool description is to be used for all double-edged turning tools.

These may be:

- Necking tools
- Double-edged circular tools
- Parting-off tools
- Double-edged contour cutting tools

Tool edge assignment

The tool description is related to **one** of the two possible tool edges. This tool edge is identified by “R1” and “+” (tool edge radius center) in the figures below. The first tool edge of the tool description is active after the tool description. The relation to an E-command is established in the first following motion block. The opposite tool edge will become effective only after edge changeover (new E-command that is different from the previous one), with the tool edge E again being assigned only on the following motion block. Now, it is possible to toggle the two tool edges as often as desired by using the two E-commands.

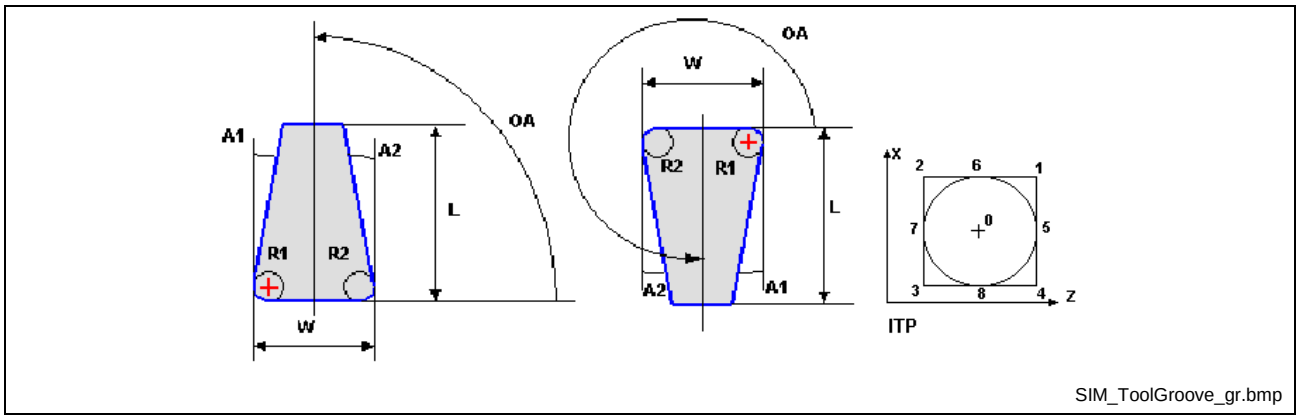
More than two tool edges E

If more than two tool edges E are used between two tool changes, the tool description must be repeated upon the third E-command, unless the third tool edge E and any further tool edge E that has not yet been used are to be assigned to the first tool edge of the description rule. After the repeated tool description, the rules for tool edge assignment will again be applicable. Refer to the examples below for a better understanding.

Note: If more than two tool edges E are used under one tool description, it is always the first tool edge of the description rule that is addressed from the third new E-word.

Note: The tool description must also be repeated, if a following new second E-word is to re-address the already active tool edge.

Syntax



L:	W	Necking width
	R1, R2	Tool edge radiuses
	A1, A2	Relief angle
	L	Length
	OA	Orientation angle
	ITP	Tool edge orientation code

Fig. 2-6: Necking tool (inside and outside)

(TOOL/GROOVE,R1,R2,L,W,A1,A2,OA,ITP)

The method of defining the tool is always the same for all mounting positions. The dimension sketch is to be turned in the proper sense to the mounting position in the description level. The orientation angle OA permits description of each and every mounting position.

The tool edge orientation code of the first tool edge is to be specified by ITP according to the description rule (1..8).

Change of the tool description orientation

If it becomes necessary to define the respectively other tool edge of the description rule as the first one, the width W must be specified with a negative sign. In this case, ITP specifies the tool edge orientation code of the new first tool edge. The positions of the tool edge radiuses and relief angles remain as they are. That means that the radius of the right-hand tool edge is still programmed with R2, as is shown in the sketch.

(TOOL/GROOVE,R1,R2,L,-W,A1,A2,OA,ITP2)

ITP2 is the orientation code of the tool edge identified by R2.

Examples

Radial necking tool width 8; the first tool edge is the left-hand one.

(TOOL/GROOVE,0.4,0.4,25,8,3,3,90,3)

Radial necking tool width 8; the first tool edge is the right-hand one.

(TOOL/GROOVE,0.4,0.4,25,-8,3,3,90,4)

Inside necking tool width 2; the first tool edge is the right-hand one.

(TOOL/GROOVE,0,0,7,2,3,3,270,1)

Axial necking tool width 6; the first tool edge is the inside one.

(TOOL/GROOVE,0.4,0.4,20,-6,3,3,0,3)

Example with more than two tool edges E

The following example illustrates programming if there are more than two tool edges E.

Task: A radial necking tool (outside) is to turn the left edge of a recess 1 with E2, then the right edge with E3. As a result of this order, the recess width W must be specified with a positive sign.

Then a further recess is to be turned with the command E4 that is to address the right-hand corner again. Since the third E-command (and any further E-command) is assigned to the first corner of the description rule in the simulation, a repeated tool descripton is required, this time with specification of a negative tool edge width.

Finally, the same right corner is to be used to machine a third recess. To keep to specific tolerances, the tool edge E1 is now to be used. Since the second E-command E1 is assigned to the opposite left-hand tool edge, another tool description becomes necessary.

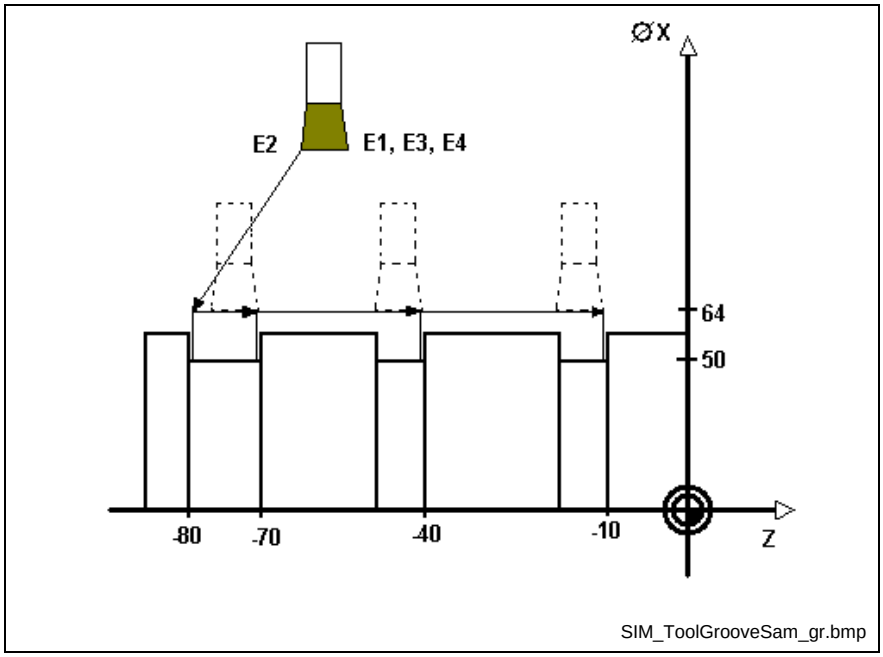


Fig. 2-7: Example with more than two tool edges E

T3 M6 ; E1 is active

(TOOL/GROOVE,0,0,20,6,3,3,90,3)

; the left-hand tool edge is the first tool edge of the tool description

E2 S200 M4

G0 X64 Z-80; 1st motion block: E2 is assigned to the left-hand tool edge

G1 X50 F0.2; necking of the left-hand edge of the first recess

G0 X64

Z-70 E3; the second E-command E3 is assigned to the right-hand tool edge

G1 X50; necking of the right-hand edge of the first recess

G0 X64

(TOOL/GROOVE,0,0,20,**-6**,3,3,90,3)

Z-40 E4; change to E4

; E4 is assigned to the 1st tool edge of the description, now the first one

G1 X50; necking of the second recess

G0 X64

; to avoid the second E-command E1 from being assigned to the

; left-hand tool, the tool description is repeated.

(TOOL/GROOVE,0,0,20,**-6**,3,3,90,3)

Z-10 E1; the second E-command E1 is assigned to the right-hand tool edge

G1 X50; necking of the third recess

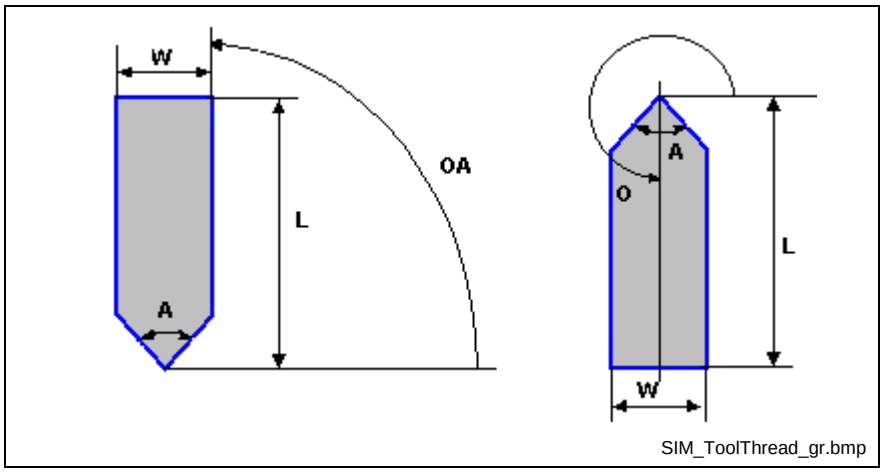
G0 X64

Thread Turning Tool

Intended use

The thread turning tool description covers simple thread turning tools for tapered, traversal and longitudinal threads. If necessary, this description type can be used for special thread turning tools (e.g. chasers).

Syntax



L: A Edge angle
W Width
L Length
OA Orientation angle

Fig. 2-8: Thread turning tool (outside and inside)

The method of defining the tool is always the same for all mounting positions.

`(TOOL/THREAD,A,L,W,OA)`

Examples

Thread turning tool, outside

(TOOL/THREAD,60,20,5,90)

Thread turning tool, inside

(TOOL/THREAD,60,20,5,270)

Tool for transversal threads

(TOOL/THREAD,60,20,5,0)

Drilling Tools

Twist Drill, Countersinking Cutter, Reamer, Threading Tap

Intended use

This description type is appropriate for all tools with a tip, a chamfer and for all purely cylindrical drilling tools:

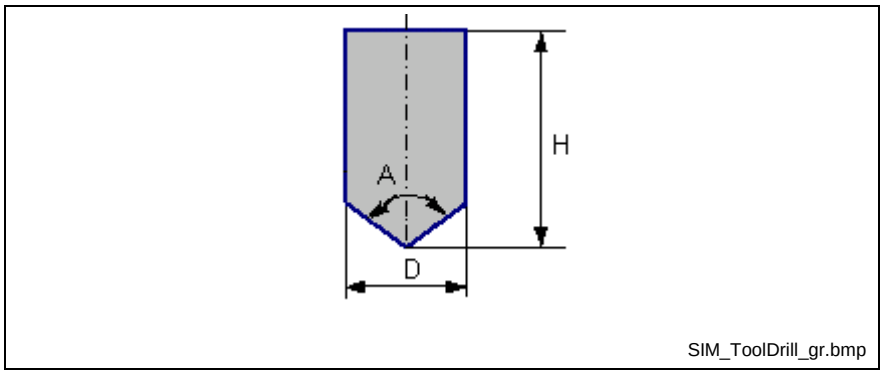
- Twist, center and disposable insert drills
- Twist, front cutting and piloted counterbores
- Reamer
- Threading taps
- As a substitution shape for boring bars and backward counterbores which do not have their own description.

Centric and driven tools

With regard to the syntax, there is no difference between stationary centric tools and driven tools.

Whereas the orientation of the tool usually follows the current length compensation direction (G-group 13 G47/G48/G49), the reverse working direction must be specified by a negative L in centric drilling processes in the left front face in the G18 plane (multi-spindle turning machine). In all cases, L must be positive. This also applies to driven tools with an opposite orientation (the tip points in the positive direction of the drill axis). If the length compensation direction is negative (G49 active), the simulator automatically changes the tool orientation.

Syntax



L: D Diameter
A Point angle
H Length

Fig. 2-9: Simple drilling tool

```
(TOOL/DRILL,D,A,H)
```

Note: The tool zero point is always at the tool tip.

Examples

Twist drill diameter 10

(TOOL/DRILL,10,118,40)

Centric twist drill diameter 20, drilling in Z+ direction

(TOOL/DRILL,20,118,-80)

Twist counterbore diameter 26

(TOOL/DRILL,26,180,100)

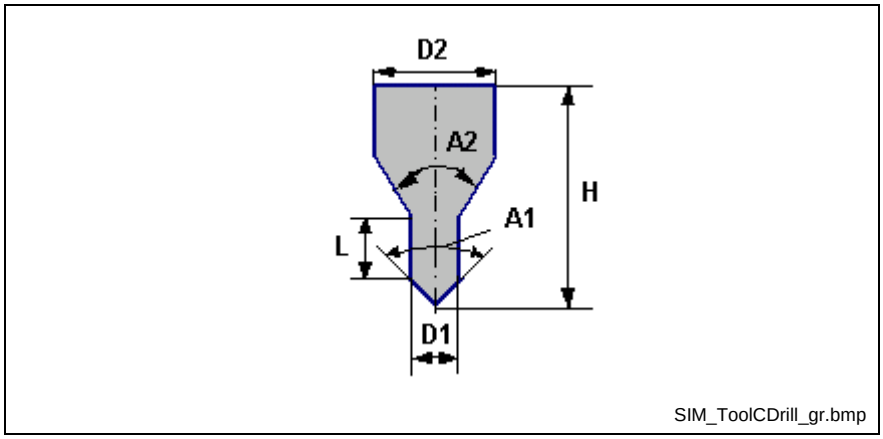
Center Drill and Step Drill

Intended use

This description shape is provided for multi-step drilling tools which, however, have their zero point always at the tip. In a future version, this description shape will also be used for boring bars and backward counterbores.

Note: The CDRILL description is not yet realized for multi-step tools in the present version. Tools that are described by CDRILL are mapped to a simple drilling tool (DRILL type).

Syntax



L:	D1	Small diameter
	D2	Large diameter
	A1,A2	Tool angle
	L	Length
	H	Total length

Fig. 2-10: Two-step drilling tool

```
(TOOL/CDRILL,D1,A1,L,D2,A2,H)
```

Examples

Center drill diameter 2.5

(TOOL/CDRILL,2.5,118,4,5,60,30)

Step drill 7/13

(TOOL/CDRILL,7,118,15,13,180,70)

Milling Tools

Cylindrical, Angle Milling and Spherical Cutters

Intended use

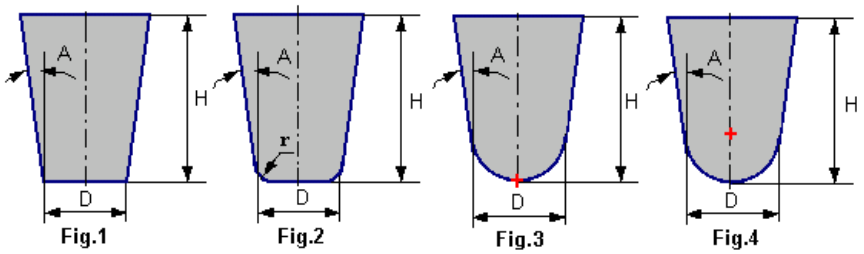
The following tool description is reserved for cylindrical and tapered milling tools, that may optionally also be provided with a round or spherical front face.

The following tools can be described:

- Shank cutter, drilling groove cutter
- Plain and end face milling cutters
- Angle mill cutter
- Side milling cutter and circular saw blades
- T-groove milling cutter
- Die milling cutter
- Cornering heads
- Measuring probes

This description shape is not inteded for several tool edges E.

Syntax



SIM_ToolMill_gr.bmp

L: D Diameter
A Taper angle
H Length
r Tool nose radius

Fig. 2-11: Cylindrical, angle milling and spherical cutters

(TOOL/MILL,D,r,H,A)

The particular shape concerned of those shown in Fig. 1-4 is defined by specification of the tool nose radius r.

Fig. 1: r=0 The tool zero point is the tool tip.

Fig. 2: r<D/2 The tool zero point is the tool tip.

Fig. 3: r=D/2 The tool zero point is the tool tip.

Fig. 4: r=-D/2 The tool zero point is the ball center.

Note: The description of angle cutting tools (A<>0) is not yet realized in the present version. At present, the tools are always described with cylindrical shape.

Note: The cutter radius compensation is always based on the half of diameter D.

Examples

Shank cutter diameter 20

(TOOL/MILL,20,0,80,0)

Angle milling cutter (V-shaped milling cutter, 45 degrees)

(TOOL/MILL,80,0,30,-45)

Cherry diameter 10, with zero point at the tip

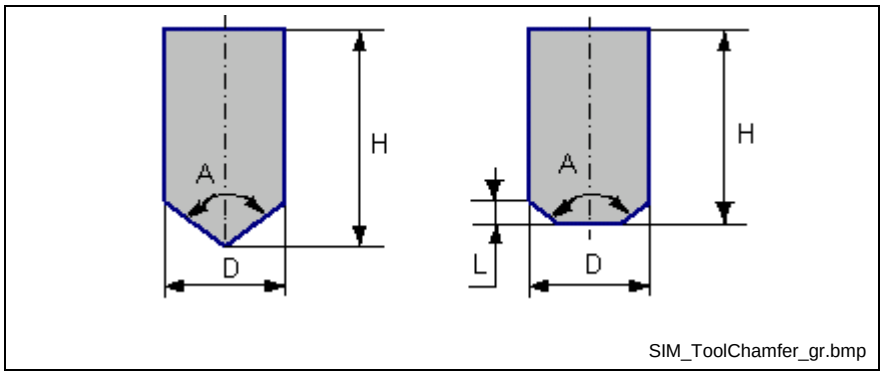
(TOOL/MILL,10,5,50,0)

Engraving Cutters and Cutters with Chamfer

Intended use

This tool shape is used for cutters with a tip (e.g. engraving cutters) and cutters with a chamfer (e.g. plain missing head).

Syntax



L: D Diameter
A Taper angle
L Chamfer length
H Length

Fig. 2-12: Cutter with tip or chamfer

Pointed tool

(TOOL/CHAMFER,D,A,H)

Tool with chamfer

(TOOL/CHAMFER,D,A,H,L)

Note: The description of cutters with chamfer is not yet realized in the present version. These tools are shown with a tip.

Examples

Plain milling head diameter 250

(TOOL/CHAMFER,250,90,70,12)

Pointed engraving cutter diameter 6

(TOOL/CHAMFER,6,90,40)

2.4 Excluding NC Blocks from the Simulation

Intended use

Some parts programs may contain instructions which cannot be processed by the interpreter of the simulation. In order to avoid any resultant error messages (syntax errors, non-initialized variable), single or several NC blocks can be excluded from simulation.

Syntax

Deactivate the simulation:

(SIMULATION/OFF)

Activate the simulation:

(SIMULATION/ON)

Note: Deactivation of the simulation is not realized in the present version.

Example

(SIMULATION/OFF)

WES 9

@51=X-OTD(,,,,1) @52=X-OTD(,,,,1) @52=X-OTD(,,,,3)

G0 G90 X=@51+5 Y=@52 Z=@53

(SIMULATION/ON)

3 Preparatory Steps for Simulation

3.1 Preparatory Steps

The preparation of an NC program for simulation comprises a number of steps which must be executed once before the graphical diagram is displayed and which provides the simulator with all input data in the expected form. The NC program is processed similarly to a download, with the following additional steps being executed:

- Excluding NC blocks from the simulation
- Adding all subroutines requires
- Inserting missing tool descriptions
- Interpreting and creating the sequence of the NC program

3.2 Excluding NC Blocks from the Simulation

This step removes all NC blocks between the NC instructions (SIMULATION/OFF) and (SIMULATION/ON).

3.3 Adding Subroutines

After completion of the NC program, all subroutines addressed by subroutine jumps (BSR instruction) and contained in the currently loaded cycle package are added.

Excluded here are a number of standard subroutines, any tool change subroutine if available, and all subroutines residing in the NC program package, program 99.

Standard subroutines

The standard subroutines are simulated internally, whereas user subroutines are simulated using the filed NC program code..

The internally simulated standard subroutines include:

.*G81,.*G82, .*G83, .*G84,.*G85,.*G86,.*G87,.*G88,.*G89,

.*G51,.*G52, .*G53,

.*G61, .*G62, .*G63, .*G64, .*G65, .*G66, .*G67, .*G68,

.*G71, .*G72, .*G73,.*G75, .*G76.

The list of the subroutine jump labels resides in the “NCPGS_NPG.INI” file (path <installation directory>\MTGUI\BasicData\Resource) in the [NPG_SIMULATION] section.

Note: Owing to the fact that the subroutines mentioned are simulated internally, an adjustment of these subroutines does not have any effect on the simulation. Use other subroutine jump labels in this case.

Tool change subroutines

The instruction for tool change execution is filed in the NCPGS_TOOLS.INI file in [SIMCOMPILER.DLL] section under the ToolChange value. This data item is used to detect a tool change when tool descriptions are inserted automatically.

Since an options dialog is not yet provided in the present version, the adjustment must be made manually if necessary. If this file

(CustomData\Resoure-Pfad) is adjusted, it is absolutely necessary to note that the applicable NC instruction should be used, and not a macro name.

Examples:

ToolChange = BSR .M6

ToolChange = MTP

The default setting of this entry is BSR .M6. If the “ToolChange” entry is used to file a subroutine jump, this subroutine will not be added to the NC program either and will, thus, not be simulated. This helps to prevent unnecessary syntax error messages.

Note: If it fails to find a subroutine jump label, the simulator continues with the following record without any error message.

3.4 Inserting Tool Descriptions

Basic Principle

Tool descriptions should be inserted immediately after the tool has been changed, i.e. before the following motion block is started. If this has not yet been made manually, the programmer is supported in two ways. This requires, however, that one or more tool changes have been written in the parts program.

The tool change is detected by means of the instruction that is declared in the NCPGS_TOOLS.INI file in the section [SIMCOMPILER.DLL] under the “ToolChange” value as tool change instruction (see Chapter 5.3 SIMCPL.INI "Relevant Settings").

If there is no tool description between two successive tool changes or between a tool change and the program end, then the simulation preparation function itself attempts to insert tool descriptions directly after the tool change. Hence, it is assumed that tool changes have been written in the NC program.

Note: Please note that tool descriptions are generated in the memory mode, i.e. the program is analyzed block by block. Program check structures and jumps are not taken into consideration here. This might result in an undesired result when tool information is inserted automatically.

Tool descriptions that are **probable** according to the processing case are inserted. This also includes some assumptions.

Note: The automatic insertion of tool descriptions is only intended to support the NC programmer. If necessary (wrong tool type or inappropriate dimensions), the tool description must be specified manually. Manually specified tool descriptions suppress the automatic insertion of the description of this tool

Generation of Tool Descriptions from GNP Cycle Parameters

If GNP cylces (Graphical NC Programming) are called up, the information on the respective tool, that is contained in the cycle parameters, is converted into a tool description and inserted directly after the tool change. Any description parameters not contained in the cycle are supplemented by reasonable assumptions.

Note: The generation of tool descriptions from GNP cycle parameters can also be used if the “DIN code acceptance” has been selected in the GNP dialog. This requires that the comment (_TOOL/…) is not deleted or changed subsequently.

Generation of Tool Descriptions from the Loaded Tool List

If GNP cycles are not called with a tool, a tool description is generated by evaluating the current preparatory functions, the spindle states and the currently loaded tool block.

The requirements for this type of insertion of tool descriptions are as follows:

- The correct tool list assigned to the NC program must be loaded in the CNC.
- The tool list must contain T-numbers.
- Each tool selected must be executed by a constant (Txxxx, but not T=@45).

Tool type definition

Definition of the tool is closely related with the NC program contents. The following table provides an overview of the criteria required for definition of a specific tool type for turning machines.

Tool type	Relation to data from tool list	Relation to data from machine parameters, simulation adjustment and NC program
Single-edged turning tool	• Correction type 3 • Tool edge orientation unequal to 0 or tool edge radius >0	• G18 • Special conditions in case of existing tool spindle (see foot note (1))
Necking tool	• Same as turning tool • There are exactly 2 different tool edges with equal radius • Length dimensions and tool edge orientations are matching each other consistently (2)	Same as single-edged turning tool
Mushroom tool	Same as single-edged turning tool, however, tool edge radius >= 3 mm	Same as single-edged turning tool
Thread turning tool	• Correction type 3 • Tool orientation >4 and radius = 0	Same as single-edged turning tool
Centric drilling tool	• Correction type 3 • Tool edge orientation 0 • Tool edge radius 0	• G18 • Workpiece spindle is rotating
Driven drilling tool	Correction type 1 or 5 **– or –** Correction type 2 or 4 with tool edge radius 0	
Step drill (driven)	• Correction type 2 • Exactly 2 tool edge corners with different lengths with equal sign are existing (2)	
Milling tool (cylindrical)	• Correction type 2 or 4 • Tool edge radius >0	
Engraving cutter	• Correction type 2 or 4 • Tool edge radius = 0	
Side milling cutter	• Correction type 3 • Tool edge radius >0	• Tool spindle existing • G18

Fig. 3-1: Tool shape relations

(1) If the machine is provided with a spindle for driven tools and the workpiece spindle is not rotating, then a turning tool is shown only if the tool edge radius is less than 8 mm; else, a milling tool is shown.

(2) Correction values for geometry and offsets are used to determine the tool type in case of multi-edged tools. Wear corrections are not taken into consideration.

If it is not possible to determine a suitable tool type, then a default tool, i.e. a single-edged turning tool, is inserted.

Tool dimensions

The tool diameter that is not defined or cannot be defined by the entry of a tool edge radius (drill correction type 1 and 3) is internally defined by the simulation. The same applies to tool edge angles and insert sizes in case of turning tools, and the like.

Some default dimensions and the default tool are residing in the “SimCpl.INI” file under the following path:

<Installation directory>\MTGUI\BasicData\Resource.

Note: Please note the following instructions, to support proper determination of the tool description.

- Avoid specifying a tool edge orientation of 0 in combination with a tool edge radius of 0, if a turning tool is concerned.
- If possible, put the spindle(s) into the operating state required before the first traversing motion after the tool change.
- Activate the interpolation plane required before the first traversing motion after the tool change.

Multi-edged tools

If several edges are assigned to a tool, with the geometric tool data being different, then it is attempted to combine two different tool edges each to one graphical diagram. Such a two-edged tool is realized in the necking tool type.

Double tools

A double tool is applicable, if two physically different tools are addressed that have one and the same tool number, but different tool edge numbers. This special case is considered in the graphical tool diagram, however only the graphical diagram pertaining to the current tool (NC block E) is visible at any time.

Tool setup list

Generation of the tool data via the setup list using the tool database is not possible in the present version.

3.5 Interpreting and Creating the Sequence of the NC Program

In the last step, the NC program is interpreted and converted into a sequence, with any program check structures (jumps, loops), if existing, being accounted for. In this step, syntax errors and impermissible expressions are detected.

The preparatory steps are completed with the creation of a 3D model and the calculation of the screen dimensions (scale for a screen-filling workpiece).

4 Capacity, Layout and Operation of the Simulation

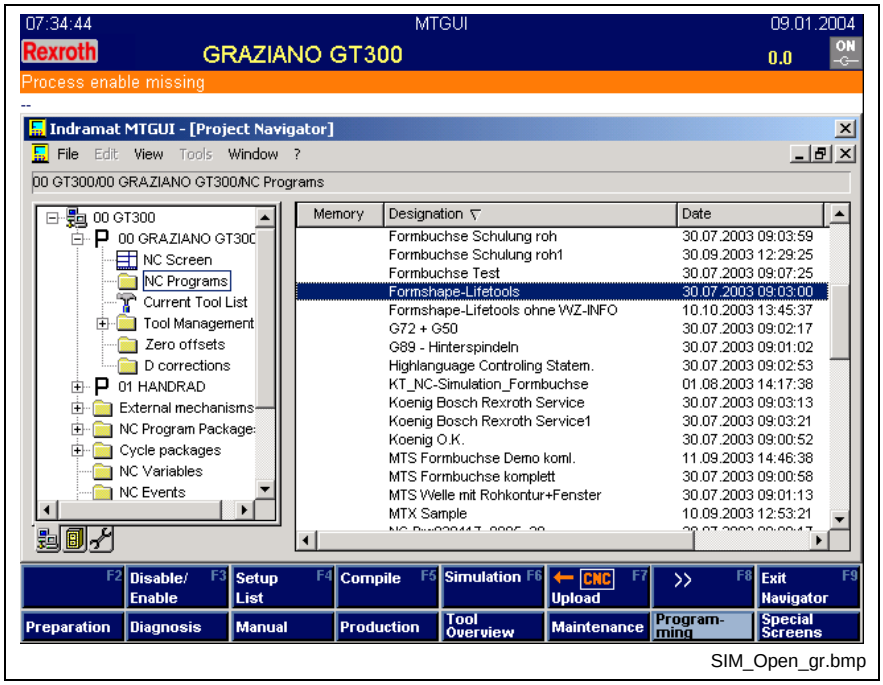
4.1 Calling the Simulation

Naviation in the project navigator

The simulation can only be started from the list view of an NC program node. To achieve this, navigate in the project navigator and to an NC program node (“free NC programs“ or within an NC program package) and select the NC program to be simulated from the list view.

<F6> Simulation

If the F-key “Simulation” is not visible, then switch to the second F-key level, if necessary, using <F9> “>>”. Then the “Simulation” key should be available. After you have pressed this key, the simulation is started only for the NC program selected.



SIM_Open_gr.bmp

Fig. 4-1: Starting the simulation

Note: The F-key “Simulation” is not shown in processes which are not provided with a tool management or at least two linear main axes.

4.2 Capacity of the Simulation

Capacity and Operating Principle of the Simulation

Essentially, the simulation is provided with the features described below. The following chapters will refer to these features and the characteristics and relations described below.

Views

The simulation provides up to 4 different views at the same time. There are 6 orthogonal views and one 3D view available.

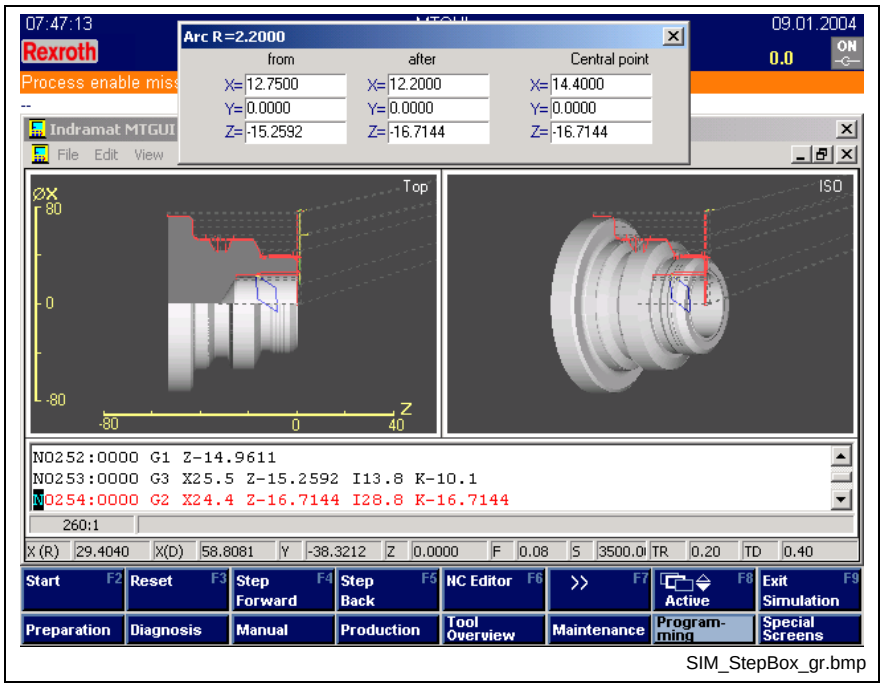
Sequential and step modes

The simulation can be operated in the **sequential mode**, that is an operating principle comparable to the automatic mode of the CNC, and in the **step mode**.

The step mode is not comparable to the automatic single block CNC mode. The simulation provides only stepwise simulation, with one step corresponding to one motion block.

The step mode can be used in the **forward and backward** directions.

In the step mode, a box appears that displays the start and target coordinates per block.



SIM_StepBox_gr.bmp

Fig. 4-2: Simulation in the step mode with coordinate box

Solid and tool path modes

In the **solid mode**, the workpiece is modelled. This mode provides the collision monitoring function and the presentation of sections. This mode requires the presence of a stock description.

In the **tool path mode**, however, there is no modelling. This mode only shows the programmed path as polyline. This mode is the only one possible without stock description.

These two modes can only be toggled via the options dialog (see "Option", p. 4-13).

Collision monitoring

The simulation monitors for possible collisions between the tool edge and the workpiece while rapid traverse motions are made. If a collision is detected, the simulation stops and a box displaying the start and target

coordinates and a corresponding message is shown. Any parts of the workpiece surface that are involved in the collision are colored separately.

The collision monitoring function is executed only in the solid mode and can also be deactivated (see "Option", p. 4-13).

Speed behavior The speed behavior corresponds with the real speed behavior at the CNC. In the tool path mode, each motion block is separated into several partial steps. The animation speed can be controlled with the numeric keys 1 (slow) to 9 (fast).
In the solid mode, the motions are not separated into partial steps. The speed cannot be changed.

Zoom The frame of each view can be changed. This is achieved via the zoom function.

Move It is also possible to move the visible frame.

The frame can also be easily moved at any time with <ALT>+<cursor key>.

Turn In addition, any workpiece in a 3D view can be turned by any angle desired.

The graphic can also be turned with <CTRL>+<cursor key>.

Sectional views Any section desired can be shown. These are half sections or full sections while the simulation is in progress. With the simulation being stopped, any sections, i.e. **axially parallel plane sections** and **radial sections** (“piece of cake”) can be defined so that the workpiece geometry (particularly inside shapes) can be evaluated.

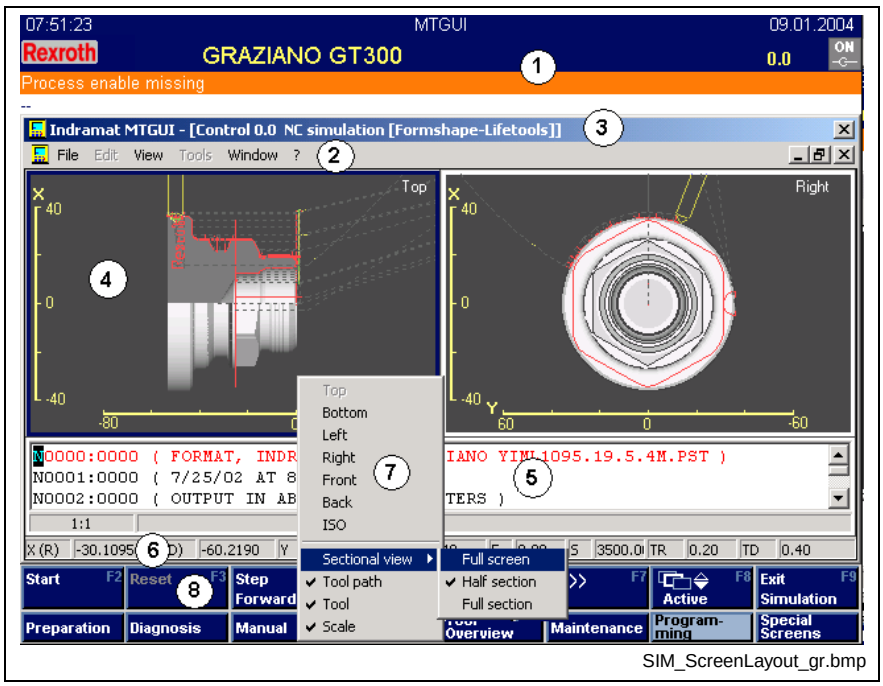
Machine design The simulation covers various configurations/designs of turning machines. These include driven tools as well as multi-spindle turning machines (spindle and counter-spindle).
The orientation of the views can be adjusted to the design of the machine. The present version covers horizontal and vertical machines (suspended spindle) “behind the turning center”.

Graphic elements Depending on the particular conditions, the graphic of each view can contain the following elements. Each element has its own colors.

- Axis scales with axis designation
- Name of the current view
- Tool (wire-frame model)
- Workpiece (3D graphic)
- Tool path for active motions (closed lines)
- Tool path for rapid traverse motions (broken lines).

4.3 Screen Structure

The screen of the simulation has the following fundamental structure.



SIM_ScreenLayout_gr.bmp

L: (1) Header
(2) Menu bar
(3) Program title: program name, control address, component name, file name
(4) Graphic window
(5) NC program window
(6) Status bar
(7) Popup menü
(8) F-key bar

Fig. 4-3: Simulation

Graphic window The graphic windows are the graphic output areas or views for the simulation. It is possible to display up to four different views at the same time. The focussed window is identified by blue borders.

NC program window The NC program window shows the compiled NC program. While the simulation is in progress, the cursor always points to the current NC block. The NC program window can also be used for interactions (e.g. find).

Status bar The status bar contains information on current states, such as coordinates, feed and spindle value.

Popup menu The popup menus can be used to make or execute window-specific settings or functions respectively. There is a separate menu for each graphic window and one for the NC program window.

4.4 Operating the Simulation

Function Key Bar

F-Key Bar of the Graphic Windows

The F-key bar of the graphic windows is available if one of the graphic windows is currently focussed. It is different from the F-key bar of the NC program window in the number of levels and the scope of functions, because the graphic window F-key bar mainly provides functions for editing the graphic.

The following overview shows the structure of the various F-key levels and the correlation between the levels.

F2	F3	F4	F5	F6	F7	F8	F9
Start/ Stop	Reset	Step forward	Step backward	NC Editor	>>	Active	Exit Simulation

Fig. 4-4: Basic F-key level, “Run” menu

<F7> >> The <F7> key moves you to the next menu level, i.e. the “Zoom” menu.

F2	F3	F4	F5	F6	F7	F8	F9
	Zoom with keys	Zoom +	Zoom -	Zoom off	>>	Active	Back

Fig. 4-5: Second F-key level, “Zoom” menu

<F7> >> The <F7> key moves you to the next menu level, i.e. the “Additional functions”.

<F9> Back <F9> returns you to the basic level.

F2	F3	F4	F5	F6	F7	F8	F9
Move	Turn	Define sections	Option	Menu	Measuring	Active	Back

Fig. 4-6: Second F-key level, “Additional functions”

<F9> Back <F9> returns you to the basic level.

<F2> Start/Stop The F2 key starts and stops the simulation in the sequential mode. The label changes according to the particular status. The function is not available at the end of the program.

<F3> Reset The Reset function returns the simulation to the first block. This also ensures that, on the next start, the previously plotted graphic will not be shown any longer. Instead, the original stock is displayed, i.e. the simulation can begin from the start.

<F4> Step forward The <F4> key, “Step forward”, simulates and stops the following motion (step mode). A box showing the start and target coordinates of the step is displayed.

<F5> Step backward The <F5> key, “Step backward”, simulates and stops the previous motion in reverse direction (step mode). “Step backward” does not update the workpiece. A box showing the start and target coordinates of the step is displayed.

<F6> NC Editor <F6> “NC Editor” exits the simulation and opens the NC program editor with the simulated NC program. Unless the cursor in the NC program window of the simulation was placed on a subroutine block, the cursor in the NC editor is positioned on the pertinent source line.

<F8> Active <F8> “Active” moves the focus among the graphic windows and the NC program window in a loop. The focus can be identified by blue borders.

Note: When, on switching on, the focus reaches the NC program window, it must be ensured that the F-key levels of the graphic windows remain as they are, except in the active basic level, the “Run” menu. Menu functions which can be used for a graphic window only are not available in this state.

Hence, the F-key level is always preserved during navigation.

<F3> Zoom with keys <F3> “Zoom with keys” moves you to a further F-key level permitting the definition of a rectangle for the frame to be magnified.

F2	F3	F4	F5	F6	F7	F8	F9
	Left	Up	Down	Right		Confirm corner point	Back

Fig. 4-7: F-key level, “Zoom with keys”

First move the cross hairs to the first corner point of the frame to be magnified with <F3> through <F6> and then confirm with <F8>. Repeat these steps for the second corner point. Click <F9> to return to the “Zoom” menu.

<F4>Magnifier + <F4> “Magnifier +” magnifies the current screen centrically..

<F5>Magnifier - <F5> “Magnifier -“ magnifies((minifies?)) the current screen centrically.

<F6> Zoom off <F6> “Zoom off” restores the original non-magnified screen.

<F2> Move <F2> “Move” moves you to a new F-key level permitting to move the frame in any direction with the help of the function keys. In addition, the “Move” cursor (cross with arrows) appears. This indicates that the mouse point has a new function.

<F7> “Reset move” undoes all moves.

F2	F3	F4	F5	F6	F7	F8	F9
	Move left	Move up	Move down	Move right	Reset move	Active	Back

Fig. 4-8: F-key level, “Move with keys”

<F3> Turn <F3> “Turn” opens a new F-key level permitting to turn the graphic (in 3D view) into any position with the help of the function keys. In addition, the “Turn” cursor (cross with arrows on spherical area) appears. This indicates that the mouse pointer has a new function.

<F7> “Reset turn” restores the graphic to its initial position.

F2	F3	F4	F5	F6	F7	F8	F9
	Turn left	Turn up	Turn down	Turn right	Reset turn	Active	Back

Fig. 4-9: F-key level, “Turn with keys”

Keys F3 through F6 are active only if the focus is on an ISO view.

<F4> Define sections

<F4> “Define sections” opens a further F-key level permitting to select a radial section and axially parallel sections. Dialog boxes permitting the definition of the sectional areas are shown for each of these section types.

F2	F3	F4	F5	F6	F7	F8	F9
Radial section	Plane section					Active	Back

Fig. 4-10: F-key level, “Define sections”

<F5> Option

<F5> “Option” opens a dialog box permitting to make global settings that are valid for all views. The new settings may have a permanent effect (when saved) or may be applicable only until the simulation is exited (when not saved).

<F6> Menu

<F6> “Menu” opens the popup menu associated with the focussed graphic or NC program window. For instance, the view of the graphic windows can be modified there.

<F7> Measuring

The <F7> “Measuring” function is under preparation at the moment. This function permits interactive measurement of areas and distances on the 3D model that has been developed.

F-Key Bar of the NC Program Window

The F-key bar of the NC program window is available if the NC program window is currently focussed. It consists of only one level whose functions are, to the major part, described above under the F-key bar for the graphic windows.

F2	F3	F4	F5	F6	F7	F8	F9
Start/ Stop	Reset	Step forward	Step backward	NC Editor	Stop at cursor position	Active	Exit Simulation

Fig. 4-11: Basic F-key level, “Run” menu

<F7> Stop at cursor position

<F7> “Stop at cursor position” starts the simulation in the sequential mode and stops it as soon as the block on which the cursor is positioned is reached. This function facilitates to selectively test parts of the NC program.

Menu Bar

The present simulation version is not provided with its own menu bar functions.

Popup Menu

Popup Menu of the Graphic Windows

The popup menu assigned to the graphic windows can be opened via the right mouse button, <shift>+<F10> or via <F6> “Menu”. To achieve this, one of the graphic windows must be focussed.

The graphic windows popup menu permits to make window-specific settings.

View Seven views are available, with the currently set view being disabled.

- Top
- Bottom
- Left
- Right
- Front
- Back
- ISO (3D view)

Sectional view The “Sectional view“ menu item opens a submenu permitting selection from

- Full frame
- Half section
- Full section

Tool path The “Tool path” menu item activates and deactivates the output of the linear graphic for the tool path. The current state is identified by the menu item being checked or not. The new state will take effect only after the simulation has been started.

The tool path is always visible (irrespective of the setting) if the parts program does not contain any stock description or if the simulation is executed in the tool path mode.

Tool The “Tool” menu item activates and deactivates the output of the tool graphic. The current state is identified by the menu item being checked or not. The new state will take effect only after the simulation has been started.

Scale The “Scale” menu item shows and hides the axis scale. The alteration will take immediate effect in the graphic. It must be noted that the “ISO” view does not provide any scale.

Popup Menu of the NC Program Window

The popup menu of the NC program window can be opened via the right mouse button or via <shift>+<F10>. To achieve this, the NC program window must be focussed.

Find The “Find” function permits to find a specific text passage in the NC program.

Start to cursor position “Start to cursor position” starts the simulation and stops it at the block of the current cursor position.

Shortcuts

The following shortcuts are provided for the described simulation functions:

Shortcuts	Simulation function
<N>	Start/Stop
<DEL>	Reset
<Shift>+<F10>	Opens the popup menu
<Ctrl>+<ALT>+<R>	Defines radial sections
<Ctrl>+>ALT>+<P>	Defines plane sections
<Ctrl>+<F>	Finds passages in the NC program window
<TAB>	Moves the focus to the next window
<Shift>+<TAB>	Moves the focus to the previous window
<ALT>+<up>	Moves the frame up
<ALT>+<down>	Moves the frame down
<ALT>+<left>	Moves the frame to the left
<ALT>+<right>	Moves the frame to the right
<Ctrl>+<up>	Turns the graphic upwards (in the ISO view only)
<Ctrl>+<down>	Turns the graphic downwards (in the ISO view only)
<Ctrl>+<left>	Turns the graphic to the left (in the ISO view only)
<Ctrl>+<right>	Turns the graphic to the right (in the ISO view only)
<ESC>	Previous F-key level, return to basic level or “crosshairs” cursor type selection
<1>...<9>	Speed in the tool path mode (slow … fast)
<Ctrl>+<F4>	Exits the simulation

Fig. 4-12: Shortcuts – simulation functions

Shortcuts	Navigation command in the NC program window
<Cursor right>	Moves the cursor to the right
<Cursor left>	Moves the cursor to the left
<Cursor down>	Moves the cursor down
<Cursor up>	Moves the cursor up
<Home>	Moves the cursor to the beginning of the line
<End>	Moves the cursor to the end of the line
<Page down>	Moves the cursor down by one page
<Page up>	Moves the cursor up by one page

Fig. 4-13: Shortcuts – navigation commands in the NC program window

Shortcuts	Block/selection command in the NC program window
<Ctrl>+<C>	Copies to the clipboard
<Shift>+<Page up>	Extends the area selected up by one page
<Shift>+<Page down>	Extends the area selected down by one page
<Shift>+<up>	Extends the area selected up by one line
<Shift>+<down>	Extends the area selected down by one line
<Shift>+<left>	Extends the area selected to the left by one line
<Shift>+<right>	Extends the area selected to the right by one line
<Shift>+<Home>	Extends the area selected up to line beginning
<Shift>+<End>	Extends the area selected up to line end
<Ctrl>+<A>	Selects all

Fig. 4-14: Shortcuts – block and selection commands in the NC program window

Zoom

The zoom is the function for magnifying and minifying the screen. The effect of zooming is different in the solid and tool path modes.

Solid mode In the solid mode, zooming is based on the magnification of the model that is cut at the zoom limits. Hence, zooming in one view always takes effect in all other views and requires computing time.

Tool path mode In the tool path mode, zooming is separately possible for the various views.

Cross hairs cursor type Within the graphic windows, there are three different functions for editing the graphic that are executed with the mouse. Each of these functions is identified by its own cursor type. The zoom function is the default function. It is activated if the cursor is of the **cross hairs** type.

Zooming with the mouse Zooming with the mouse is the first possibility of magnifying a frame. Place the cursor on the first corner point of the desired frame (rectangle) and, while holding the left mouse button, move it to the second corner point. Release the mouse button to compute the new screen.

Zooming with keys Zooming with keys executes the same function as zooming with the mouse and can be used if a mouse is not available. This function has been described above, for the graphic windows F-key bar.

Magnifier +/- The magnifier + and – permits to efficiently magnify and minify the screen in relation to the center of the screen. These F-key functions are recommended for zooming, if a mouse is not available.

Move

The “Move” function permits to modify the visible frame without having to change the scale.

Move cursor type

A further function of the mouse pointer is indicated by the **“Move” cursor type,** a cross with arrows.

Moving with the mouse

The “Move” cursor type is shown if <F2> “Move” has been selected. Moving the mouse pointer also moves the frame into the direction desired. <ESC> or <F9> returns the cursor type to the cross hairs cursor.

Moving with keys

<F2> “Move” also permits moving with the function keys. The “Moving with keys” F-key level that appears provides its own key for each direction.

Moving with shortcuts

Moving with shortcuts permits to move the frame independently of the current cursor type. To achieve this, the <ALT> key must be pressed together with the cursor keys.

Turn

The “Turn” function permits turning a 3D view into any position desired.

Turn cursor type

Turning is the third function of the mouse pointer in the graphic windows which is indicated by its own **“Turn” cursor type.** This type is identified by a cross with arrows placed on a spherical area.

Turning with the mouse

The “Turn” cursor type is shown if <F3> “Turn” has been selected. Moving the mouse pointer turns the workpiece. <ESC> or <F9> returns the cursor type to the cross hairs cursor.

Turning with keys

<F3> “Turn” also permits turning with the function keys. The “Turning with keys” F-key level that appears provides its own key for each direction.

Turning with shortcuts

Turning with shortcuts permits to turn the frame independently of the current cursor type. To achieve this, the <CTRL> key must be pressed together with cursor keys.

Define Sections

General Notes on Sectional Views

The freely definable sections that are treated here do not correspond to the “Sectional view” setting of the popup menu. Whereas in the popup menu the half and full sections are specified window by window and this setting always remains active during the simulation, the freely definable sections are intended to view details with the simulation stopped. These sections take effect in all views and will be lost on zooming. They superimpose the “Sectional view” settings of the popup menu. Radial and plane sections are also superimposed.

Radial Sections

A radial section removes an angular segment of the workpiece (specified by an angle about the longitudinal axis).

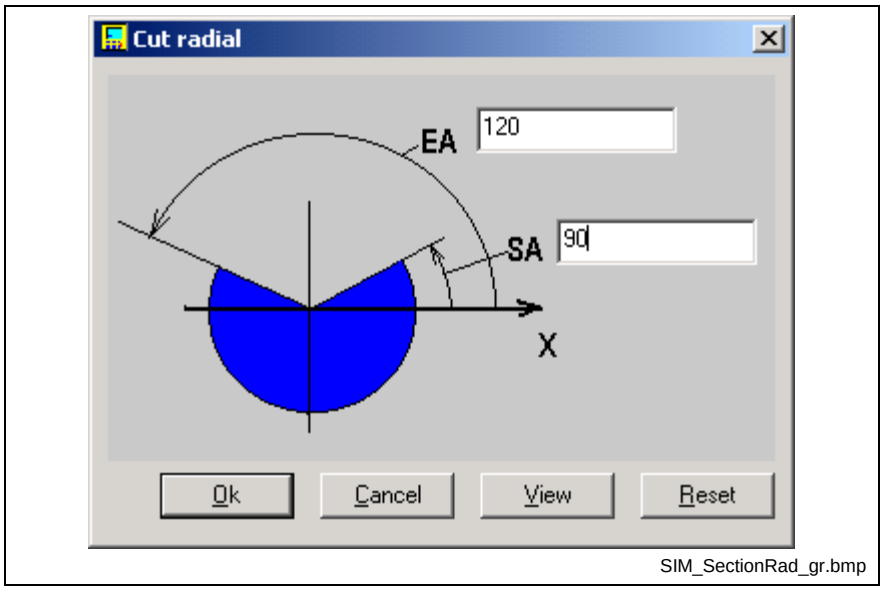


Fig. 4-15: Dialog for defining radial sections

EA, SA EA and SA are the input fields for the angles of cut.

Ok The Ok button or the <enter> key confirms the angle specifications and closes the dialog.

Cancel “Cancel” or <ESC> closes the dialog without saving the changes to the sections.

View” “View” makes any settings immediately visible without closing the dialog.

Reset “Reset” deletes the radial section.

Plane Sections

Plane sections are axially parallel sections. The sectional areas must be defined by coordinates in the X, Y and/or Z axes. Since each of the three sections cuts the workpiece in two parts, the user must define the one to be shown.

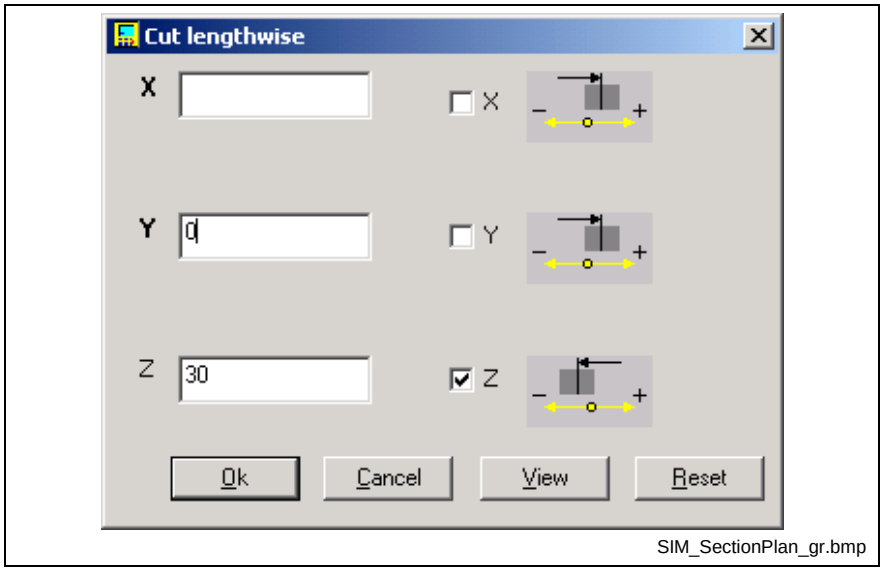


Fig. 4-16: Dialog for defining radial sections

X, Y, Z X, Y and Z are the input fields for the coordinates of the axially parallel sectional areas. The cuts are only made in the planes in which the associated coordinate has been defined. The toggle buttons permit to select the respectively other part for being visible. The symbol to the right illustrates the viewing angle of the sectional area that becomes active after a part has been removed.

Ok The Ok button or the <enter> key confirms the plane sections and closes the dialog.

Cancel “Cancel” or <ESC> closes the dialog without saving the changes to the sections.

View “View” makes any settings immediately visible without closing the dialog.

Reset “Reset” deletes the axially parallel sections.

Option

The “Option” function permits to make general and global settings for the simulation, with the global settings being applicable to all windows.

The presently available dialog is only provisional and in English language only.

At present, further settings can only be edited manually in the INI files (see Chapter 5, “Simulation Adjustment”).

The settings will take effect only after the simulation and after start of the program.

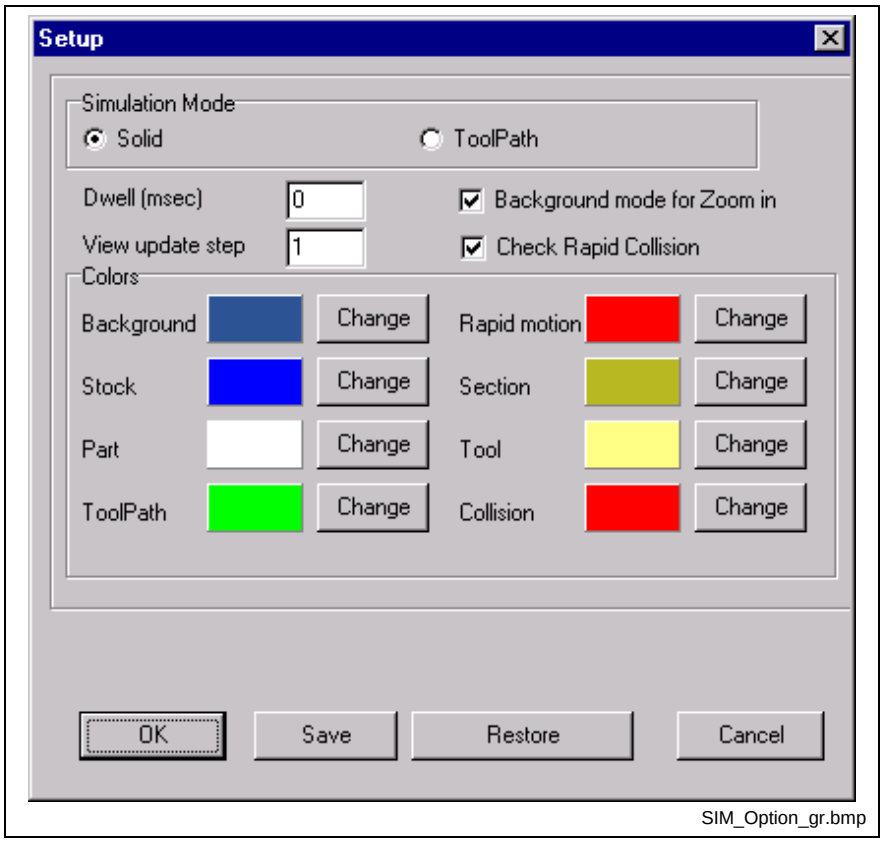


Fig. 4-17: Setup dialog

Note: The user should only change the settings described below.

Simulation Mode The “Solid” radio button activates the solid mode, “ToolPath” the tool path mode. This function is disabled if no stock has been defined.

Check Rapid Collision The “Check Rapid Collision” toggle button activates and deactivates the collision monitoring function.

Colors “Change” permits to change the color setting of the following graphic elements/areas:

- Background
- Stock: areas of stocks
- Part: areas processed
- ToolPath polyline for feed motions
- Rapid motion polyline for rapid traverse motions
- Section sectional areas
- Tool caution: complementary model!
- Collision areas affected by a collision

OK The OK button confirms the settings and closes the dialog.

Save “Save” saves the settings. They will take effect again when the simulation is restarted.

Restore “Restore” activates the saved changes.

Cancel “Cancel” closes the “Setup” dialog without saving the settings made.

5 Simulation Adjustment

5.1 SimConfig_xxx_yyy.INI

File Meaning and Origin

Meaning and origin

The file described below contains all settings of the simulation as well as any relevant information obtained from the machine parameter set for the particular process concerned. This file is automatically created and completed with a number of presettings by the MTGUI. The file is updated after the machine parameter set has been changed. Since some assumptions are made when this file is created or updated, certain machine configurations may require manual revision. The same applies for simulation options which are not yet provided with their own dialog.

Path

The file resides under the following path: <Installation directory>\MTGUI\Project_zzz\CustomData\Resource, file name SimConfig_xxx_yyy.INI (with xxx being the device number and yyy being the process number).

Content

System Parameters

Tool management

Meaning:

The simulation basically requires an existing tool management.

Rexroth MTC 200 parameter set:

A00.052 = 0 no
A00.052 = 1 yes

SimConfig-File:

Section	[PARAMETER]
Address	ToolManagement
Values	0/1

Number of tool edges

Meaning:

Used for the calculation of tool shapes. (Applicable only if the tool management is available.)

Rexroth MTC 200 parameter set:

A00.054 = 1..9

SimConfig-File:

Section	[PARAMETER]
Address	MaxEdges
Values	1..9

Tool technology

Meaning:

Defines whether the turning/milling or grinding technology is available. The simulation is not available for grinding. (This parameter is applicable only if the tool management is available.)

Rexroth MTC 200 parameter set:

A00.091 = 0 (turning/milling) or 1 (grinding)

SimConfig-File:

Section	[PARAMETER]
Address	ToolTechnology
Values	0/1

Technology

Meaning:

This parameter is calculated from several machine parameters and defines whether the machine concerned is a turning or a milling machine.

SimConfig-File:

Section	[PARAMETER]
Address	Technology
Values	Milling / Lathe

Number of NC axes

Meaning:

Number of NC axes that can be incorporated in the simulation of this process, including dynamically assigned axes. The number is unequal to the physical axis number of the CNC.

SimConfig-File:

Section	[PARAMETER]
Address	MaxIndexAxes
Values	>= 0

Process Parameters

Default length measuring unit

Meaning:

Defines the length measuring unit used by the NC program when it starts.

Rexroth MTC 200 parameter set:

Bxx.001 = 0	mm
Bxx.001 = 1	inch

InchSimConfig-File:

Section	[PARAMETER]
Address	BasicUnit
Values	mm / inch

Displayed decimal places for distances

Meaning:

Format of the display of coordinate values.

Rexroth MTC 200 parameter set:

Bxx.003 = 0 4

SimConfig-File:

Section	[PARAMETER]

Address PrecisionDisplay
Values 0...4

Default interpolation plane

Meaning:
Defines the interpolation plane set after the start of the NC program.

Rexroth MTC 200 parameter set:
Bxx.004 = 0/1/2 for G17/G18/G19

SimConfig-File
Section [PARAMETER]
Address BasicPlane
Values G17/G18/G19

Default radius/diameter programming

Meaning:
Represents the start requirement for scaling of the X-axis.

Rexroth MTC 200 parameter set:
Bxx.013 = 0 G15 (radius)
Bxx.013 = 1 G16 (diameter)

SimConfig-File:
Section [PARAMETER]
Address BasicXScale
Values G15/G16

Cartesian-polar coordinate transformation

Meaning:
Specifies whether coordinate transformation (polar coordinates and cylinder coordinates) is permitted.

Rexroth MTC 200 parameter set:
Bxx.035 = 0 no
Bxx.035 = 1 yes

SimConfig-File:
Section [PARAMETER]
Address Transformation
Values 0/1

Default tool length correction

Meaning:
Start requirement for an NC program. Affects the orientation of drilling and milling tools.

Rexroth MTC 200 parameter set:
Bxx.038 = 0/1/2 for G47/G48/G49

SimConfig-File:
Section [PARAMETER]
Address BasicLengthCorr
Values G47/G48/G49

Default rotary axis starting logic

Meaning:
Start requirements for an NC program with set rotary axis logic.

Rexroth MTC 200 parameter set:

Bxx.056 = 0/1/2 for G36/G37/G38

SimConfig-File

Section	[PARAMETER]
Address	BasicRALogic
Values	G36/G37/G38

Axis Parameters

Process allocation

Meaning:

Reflects the NC axis assignment to processes. The first entry represents the basic state.

Rexroth MTC 200 parameter set:

Cxx.000 prozessnummer[,prozessnummer1 [,...]]

SimConfig-File:

Section	[AXIS_XXX] XXX is the index of the axis
Address	Process
Values	prozessnummer[,prozessnummer1 [,...]]

Axis type

Meaning:

Defines the type of axis movement (translation, rotation).

Rexroth MTC 200 parameter set:

The axis type is contained in the system parameters.

SimConfig-File:

Section	[AXIS_XXX] XXX is the index of the axis
Address	Type
Values	ANALOG_LINEAR_AXIS

ANALOG_ROTARY_AXIS
ANALOG_MAIN_SPINDLE
ANALOG_COMB_TURRET_AXIS
C_AXIS
DIGITAL_LINEAR_AXIS
DIGITAL_ROTARY_AXIS
DIGITAL_MAIN_SPINDLE
DIGITAL_COMB_TURRET_AXIS
DIGITAL_C_AXIS

Axis designation

Meaning:

First axis designation.

Rexroth MTC 200 parameter set:

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

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	Name
Values	X,Y,Z,U,V,W,A,B,C,S with/without index 1,2,3

Axis meaning (axis functions)

Meaning:

Lists the possible axis meanings (positions in the axis compound).

Rexroth MTC 200 parameter set:

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

Up to 4 meanings are possible (separated by a comma).

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number.
Address	Function
Values	X,Y,Z,U,V,W,A,B,C,S(S1), S2, S3 (e.g.: S1,Y)

Graphic axis functions

Meaning:

Describes the direction of installation of an NC axis. Is calculated from various parameters and must be corrected, if necessary.

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number.
Address	GraphFunction
Values	X,Y,Z for linear axes, A,B,C for rotary axes and spindles

Tool spindle

Meaning:

This parameter describes whether the spindle is a workpiece spindle or a tool spindle. It is calculated from various parameters and must be corrected, if necessary (applicable to the spindle only).

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	ToolSpindle
Values	1 tool spindle, 0 workpiece spindle

Axis designation for rotary axis mode

Meaning:

Axis designation in the rotary axis mode (only applicable to digital C-axes.)

Rexroth MTC 200 parameter set:

Cxx.055 = X,Y,Z,U,V,W,A,B,C with/without index 1,2,3

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number.
Address	NameRA
Values	X,Y,Z,U,V,W,A,B,C with/without index 1,2,3

Number of allocated rotary axis

Meaning:

Refers to the index of a rotary axis that is assigned to a spindle and is addressed in the rotary axis mode (only applicable to main spindles).

Rexroth MTC 200 parameter set:

Cxx.064 = physical axis number / 0 no axis assigned

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	AllocRA
Values	0 no axis, >0 index of the allocated axis

Second axis designation

Meaning:

Axis name with the coordinate transformation G31/G32.

Rexroth MTC 200 parameter set:

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

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	Name2
Values	X,Y,Z,U,V,W,A,B,C with/without index 1,2,3

Reversal with transformation

Meaning:

If reversal is selected, the particular rotary axis rotates in opposite direction if coordinates are transformed.

Rexroth MTC 200 parameter set:

Cxx.076 = 0 normal
Cxx:077 = 1 reversal

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	TransDir
Values	0/1

Graduations per revolution

Meaning:

This parameter defines the scaling of a rotary axis by specifying the units per revoluation (only applicable to rotary axes and combined turret axes). Spindles always have 360 units per revolution.

Rexroth MTC 200 parameter set:

Cxx.006 = number

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	Division
Values	Number

Positive travel range limit

Meaning:

This measure can be used for screen dimensioning where linear axes are concerned. If the positive and the negative travel range limits are the same in case of rotary axes, then the axis is not limited.

Rexroth MTC 200 parameter set:

Cxx.011 = number unit

The unit may be “units” for rotary axes or “mm” / “inch” for linear axes.

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	TravelLimPos
Values	Value mm/inch/units

Negative travel range limit

Meaning:

This measure can be used for screen dimensioning where linear axes are concerned. If the positive and the negative travel range limits are the same in case of rotary axes, then the axis is not limited.

Rexroth MTC 200 parameter set:

Cxx.012 = number unit

The unit may be “units” for rotary axes or “mm” / “inch” for linear axes.

SimConfig-File

Section	[AXIS_XXX] XXX is the index number
Address	TravelLimNeg
Values	Value mm/inch/units

Maximum velocity

Meaning:

Can be used for calculating the velocity.

Rexroth MTC 200 parameter set:

Cxx.016 = value unit

The unit may be “units/min” for rotary axes or “mm/min” or “inch/mm” for linear axes.

SimConfig-File

Section	[AXIS_XXX] XXX is the index number
Address	MaxSpeed
Werte	Value mm/min . inch/min units/min

Maximum acceleration rate

Meaning:

Maximum acceleration in the spindle mode. Can be used for calculating the velocity (only applicable to main spindles).

Rexroth MTC 200 parameter set:

Cxx.056 = value
Unit: Rad/s²

SimConfig-File:

Section	[AXIS_XXX] XXX is the index number
Address	MaxAccel
Values	Value

Maximum speed

Meaning:

Maximum speed in the spindle mode. Can be used for calculating the velocity (only applicable to main spindles).

Rexroth MTC 200 parameter set:

Cxx.049 = value
Unit: 1/min

SimConfig_File

Section	[AXIS_XXX] XXX is the index number
Address	MaxRevolution
Values	Value

Simulation Options

Number of lines in the NC program window

Meaning:

Adjusts the number of visible NC program lines in the NC program window. The default setting is 4.

SimConfig-File:

Section	[LINE]
Address	Number
Values	>0

Number of graphic windows

Meaning:

Defines the number of graphic windows. The default setting is 2.

SimConfig-File:

Section	[VIEW]
Address	Number
Values	1, 2 or 4

Axis orientation

Meaning:

This setting specifies the orientation of the first axis of the top view. If turning machines are concerned, this is the positive direction of the Z-axis; if milling machines are concerned, it is the positive direction of the X-axis. This setting permis to adjust the simulation to various machine designs (horizontal machine, vertical machine, etc.).

SimConfig-File:

Section	[AxisOrientation]
Addresse	Direction
Values	0..3 0 to the right, 1 upwards, 2 to the left, 3 downwards

Before the turning center

Meaning:

This setting specifies whether the machine concerned is a turning machine “before the turning center”. In this case, the orientation of the second axis of the top view is offset by 180 degrees (only applicable to turning machines; not evaluated at present).

SimConfig-File:

Section	[AxisOrientation]
Address	Site
Values	0 behind turning center, 1 before turning center

Graphic Window Settings

Sectional view

Meaning:

Saves the previously set sectional view type.

SimConfig_File:

Section	[VIEWx] with x being the graphic window index 0..3
Address	Sectioning
Values	NoSection/HalfSection/FullSection

Scale representation

Meaning:
Saves the previous state for representation of scale on/off.

SimConfig_File:

Section	[VIEWx] with x being the graphic window index 0..3
Address	ShowScale
Values	On/Off

Tool presentation

Meaning:
Saves the previous state for representation of tool on/off.

SimConfig_File:

Section	[VIEWx] with x being the graphic window index 0..3
Address	ShowTool
Values	On/Off

Tool path presentation

Meaning:
Saves the previous state for representation of tool path on/off.

SimConfig_File:

Section	[VIEWx] with x being the graphic window index 0..3
Address	ShowToolPath
Values	On/Off

View presentation

Meaning:
Saves the previously selected view.

SimConfig_File:

Section	[VIEWx] with x being the graphic window index 0..3
Address	View
Values	Top/Bottom/Front/Rear/Right/Left/ISO

Example

The section below gives an example of a turning machine with Y-axis and driven tools.

```
 [PARAMID]
; Ident information
Number = 03
Name = Niles N20
Size = 1234
Date = 19.09.99
Time = 09:09:09
Device = 0
Process = 0

[PARAMETER]
ToolManagement = 1
MaxEdges  = 4
```

```
MaxIndexAxes = 5
ToolTechnology = 0
Technology = Lathe

BasicUnit = mm
PrecisionDisplay = 4
BasicPlane = G18
BasicXScal = G16
Transformation = 1
ToolLengthCorr = G48
BasicRALogic = G36

[AXIS_001]
Process = 0
Type = DIGITAL_LINEAR_AXIS
Name = X
Function = X,Z
GraphFunction = X
Name2 = Y1
TransDir = 0
TravelLimPos = 1000.0 mm
TravelLimNeg = -100.0 mm
MaxSpeed = 10000 mm/min

[AXIS_002]
Process = 0
Type = DIGITAL_LINEAR_AXIS
Name = Y
Function = Y
GraphFunction = Y
TransDir = 0
TravelLimPos = 100.0 mm
TravelLimNeg = -100.0 mm
MaxSpeed = 2,000 mm/min

[AXIS_003]
Process = 0
Type = DIGITAL_LINEAR_AXIS
Name = Z
Function = Z,X
GraphFunction = Z
Name2 = Z
TransDir = 0
TravelLimPos = 1000.0 mm
TravelLimNeg = 50.0 mm
MaxSpeed = 10000 mm/min
```

```
[AXIS_004]
Process = 0
Type = DIGITAL_C_AXIS
Name = S
Function = S,Y
GraphFunction = C
ToolSpindle = 0
NameRA = C
Name2 = Y1
TransDir = 0
TraveLimPos = 0
MaxRevolution = 3000
MaxAccel = 200

[AXIS_005]
Process = 0
Type = DIGITAL_MAIN_SPINDLE
Name = S2
Function = S2
GraphFunction = C
ToolSpindle = 1
AllocRA = 0
MaxRevolution = 5000
MaxAccel = 500

[LINE]
Number = 3

[VIEW]
Number = 2

[AxisOrientation]
Direction = 0
Site = 0

[VIEW0]
Sectioning     = NoSection
ShowScale      = Off
ShowTool       = On
ShowToolPath   = On
View           = Top

[VIEW1]
Sectioning     = NoSection
ShowScale      = Off
ShowTool       = On
ShowToolPath   = On
```

View = ISO

[VIEW2]
Sectioning = NoSection
ShowScale = Off
ShowTool = On
ShowToolPath = On
View = Top

[VIEW3]
Sectioning = NoSection
ShowScale = Off
ShowTool = On
ShowToolPath = On
View = ISO

5.2 NCPGS_TOOLS.INI

File Meaning and Origin

Meaning and origin The file described below contains settings which must be saved for the NC programming system. The file is created in the NC programming system in the course of various processes and/or operator steps.

Path The file resides in the following path: <Installation directory>\MTGUI\Project_zzz\CustomData\Resource, file name NCPGS_TOOLS.INI.

Editing instructions A file of the same name also resides under <Installation directory>\MTGUI\BasicData\Resource. If sections are added in CustomData, the complete existing content of the section must first be inserted from BasicData.

Relevant Settings

Tool change command Meaning:

Entering a tool change command is essential for automatic insertion of tool information. The NC command must be entered, not the macro. The default setting is “BSR .M6”.

NCPGS_NPG.INI-File:
Section [SIMCOMPILER.DLL]
Address ToolChange
Value NC instruction for tool change (e.g. BSR .TCHG)

5.3 SIMCPL.INI

File Meaning and Origin

Meaning and origin

The SIMCPL.INI file contains basic tool data for the preparatory simulation steps. If it is not possible to determine any values from the program or the tool data, then these default values are used.

In the present version, the default values cannot be adjusted (overlaid) via a duplicate of the file in the Customdata\Resource path.

Path

<Installation directory>\MTGUI\BasicData\Resource\SimCpl.ini

Relevant Settings

The section below lists the default values on delivery. The length unit is mm.

Section **[TOOL_BASE_SETTINGS]**

Address STANDARD_FINISH_BA
Value 35
Meaning Tool edge setting angle for default turning tool (finishing cycle)

Address STANDARD_ROUGH_BA
Value 15
Meaning Tool edge setting angle for default turning tool (roughing cycle)

Address STANDARD_FINISH_A
Value 50
Meaning Tool edge angle for default turning tool (finishing cycle)

Address STANDARD_ROUGH_A
Value 70
Meaning Tool edge angle for default turning tool (roughing cycle)

Address STANDARD_FINISH_IC
Value 10
Meaning Insert size for default outside turning tool (finishing cycle)

Address STANDARD_ROUGH_IC
Value 10
Meaning Insert size for default outside turning tool (roughing cycle)

Address STANDARD_FINISH_IC_IN
Value 6
Meaning Insert size for default inside turning tool (finishing cycle)

Address STANDARD_ROUGH_IC_IN
Value 6
Meaning Insert size for default inside turning tool (roughing cycle)

Address BUTTON_OA
Value 45
Meaning Shank angle for mushroom tool

Address GROOVE_A1
Value 3
Meaning Undercut angle for recessing tool (1st tool edge)

Address GROOVE_A2
Value 3
Meaning Undercut angle for recessing tool (2nd tool edge)

Address DRILL_D
Value 10
Meaning Diameter for drill

Address DRILL_A
Value 118
Meaning Point angle for drill

Address CDRILL_D1
Value 5
Meaning Drill diameter for center drill

Address CDRILL_A1
Value 118
Meaning Point angle 1 for center drill

Address CDRILL_D2
Value 15
Meaning Shank diameter for center drill

Address CDRILL_A2
Value 150
Meaning Point angle 2 for center drill

Address MILL_D
Value 1
Meaning Diameter for milling cutter

Address MILL_A
Value 0
Meaning Point angle for milling cutter

Address	MILL_R
Value	0
Meaning	Radius for milling cutter

Address	MILL_L
Value	10
Meaning	Shank length for milling cutter

Address	THREAD_A
Value	60
Meaning	Point angle for thread turning tool

Address	THREAD_W
Value	5
Meaning	Width for thread turning tool

Address	CHAMFER_D
Value	4
Meaning	Diameter for engraving cutter

Address	CHAMFER_A
Value	90
Meaning	Point angle for engraving cutter

Address	DEFAULT_TOOL
Value	(TOOL/STANDARD,35,50,0.4,10,3)
Meaning	This tool description is used only if it was not possible to determine a tool shape.

6 Index

P

R

S

T

U

V

Z

7 Service & Support

7.1 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**

7.2 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**

7.3 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.

7.4 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.

7.5 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 NV/SA
Henri Genessestraat 1
1070 Bruxelles

Tel: +32 (0) 2 582 31 80
Fax: +32 (0) 2 582 43 10
info@boschrexroth.be
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 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

Switzerland West - Schweiz West

Bosch Rexroth Suisse SA
Av. Général Guisan 26
1800 Vevey 1

Tel.: +41 (0)21 632 84 20
Fax: +41 (0)21 632 84 21

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 34630 Sefaköy Istanbul Tel.: +90 212 541 60 70 Fax: +90 212 599 34 07	Turkey - Türkei Servo Kontrol Ltd. Sti. Perpa Ticaret Merkezi B Blok Kat: 11 No: 1609 80270 Okmeydani-Istanbul Tel: +90 212 320 30 80 Fax: +90 212 320 30 81 remzi.sali@servokontrol.com www.servokontrol.com	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	USA Central Region - Mitte	USA Southeast Region - Südwest	**USA SERVICE-HOTLINE**
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	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	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	- 7 days x 24hrs - **+1-800-REX-ROTH** **+1 800 739 7684**
USA East Region – Ost	USA Northeast Region – Nordost	USA West Region – West	
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	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	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	Canada West - Kanada West	Mexico	Mexico
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	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	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	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	Argentina - Argentinien	Brazil - Brasilien	Brazil - Brasilien
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	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)	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	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

R911298710

Printed in Germany
DOK-MTC200-NC**SIM*V23-AW01-DE-P