engineering
mannesmann
Rexroth

MTC200/MT-CNC
NC Program Restart
18VRS

Application Manual

DOK-MTC200-SATZVOR*V18-ANW1-EN-P

282405

Indramat

Title MTC200/MT-CNC NC Program Restart 18VRS

Type of document Application Manual

Document code DOK-MTC200-SATZVOR*V18-ANW1-EN-P

Internal file reference
- Folder 5 / Register 11
- Drawing number: 109-1041-4116-01

Purpose of the document This document is used for support during NC program restart commissioning.

Configuration control

Revision	Date	Remarks
109-1041-4116-01	07.98	New issue

Published by INDRAMAT GmbH • Bgm.-Dr.-Nebel-Str. 2 • D-97816 Lohr a. Main

Phone 09352/40-0 • Tx 689421 • Fax 09352/40-4885

Dept. ESM2 (JA)

Contents

1 Introduction

Task of NC Program Restart

NC program restart is used for establishing the states in the controller and at the machine that are required for starting the machining process at any block in the NC program.

2 Operator Input Sequence

2.1 Requirements

Operator Input

- The 'ready to start' signal (PxxSREADY=1) of the process is applied.
- The controller contains the program that shall be executed.
- The program that shall be executed is selected.
- Automatic mode has been selected.
- No program is active (PxxSACTIV=0)

Parameter value assignment

The ´reverse program execution required´ (Bxx.033) and ´manual axis jogging causes reset´ (Bxx.036) process parameters must be set to NO.

Key assignments

The ´NC program restart´ function is not included in the GUI standard assignments of the F and M keys. The machine manufacturer must commission NC program restart according to the description below.

2.2 Selecting NC Program Restart

Selection

To select NC program restart, the operator actuates the associated machine control key.

Screen

The user interface then displays the NC program restart screen (see figure below), including the associated machine control bar.

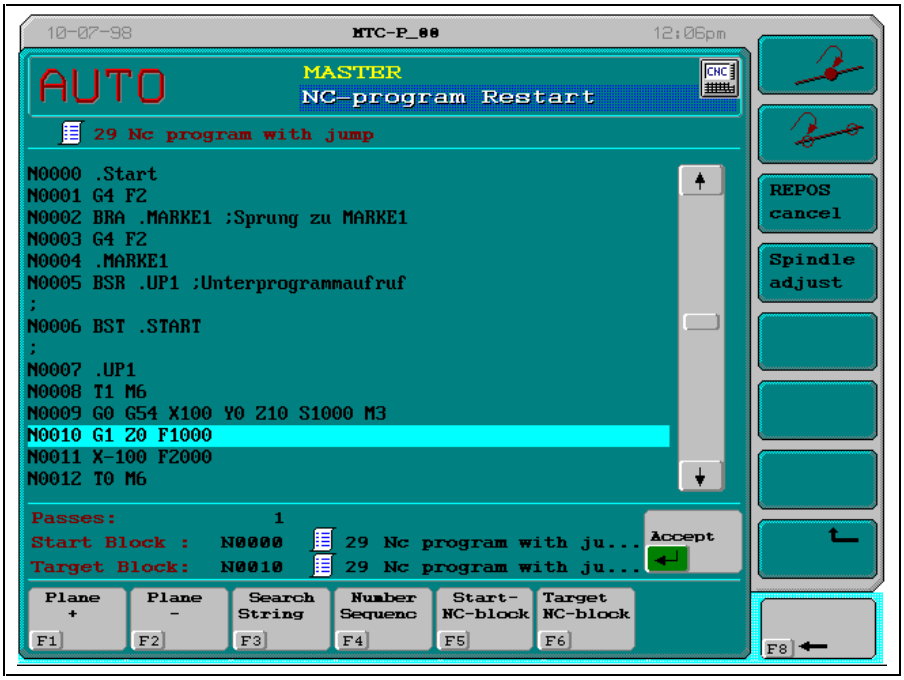

Figure 2-1: NC program restart screen (selecting the target block)

General functionality

While the NC program restart is active, the operator can use the <Ctrl> <Page Up> or <Ctrl> <Page Down> key combination to select a different station (provided that more than one station exists and the ´NC program restart´ function has been installed there).

2.3 Selecting a Target Block

Target block

The target block is the NC block from which the NC continues machining. The NC does not interpret this block during the computing process. The target block can be a primary or a secondary block.

A primary block is a block that contains all the words (NC commands) that are required for starting the machining sequence there. A primary block is identified by the primary block character ´**:**´. All the other blocks are referred to as secondary blocks.

Selection

The target block is selected in the same fashion as during block search:

- Use the cursor keys to select the target block, and press the <Enter> key to accept it.
- Use the <POS 1> key to jump to the block N0000. Use the <End> key to jump to the block before the end of the program. Use the page key to scroll an NC program that exceeds the display capacity of one screen page.
- The <Plane+> function locates the jump label that is related to a jump instruction, and sets the cursor to the corresponding NC block.
- Pressing the <Plane-> key enables the operator to return from a subroutine to the calling point.
- Using the <Search String> function, you can locate any character string in the direction towards the end of the displayed NC program (forward direction).The character string may also be a part of a comment.
- Press the <Enter> key to accept the required target block.

2.4 Selecting the Start Block

Start block

The start block is the NC block from which onwards the NC shall start the computing process. This can either be the first NC block of the NC program or the last primary block before the target block.

Selection

Using the <NC block restart from last primary block> softkey on the user interface to select the last primary block before the target block as the start block.

If you do not select this softkey, the NC always starts the computing process at the beginning of the NC program.

Note: NC program restart from the last primary block should be used for applications, in particular, for which the controller and/or machine states could not be determined during the computing process (e.g. applications with background magazines).

Start block selection is mandatory if there are subroutines at the beginning of an NC program. Otherwise, the NC would start the computing process with the first block of the subroutine.

When the NC program restart screen is exited, the NC sets the 'NC program restart active' interface signal. This signals the SPS that the 'NC program restart' submode is active and that the computing process can be started.

2.5 Peculiarities in the Selection of Start and Target Block

Absolute jumps to the labels BRA, BST, BEQ, BNE, BPL, BMI, BRF

If the cursor is placed on an NC block that contains a jump command

BRA, BST, BEQ, BNE, BPL, BMI, BRF,

pressing <Plane+> causes the label that is specified in the jump command to be located automatically, and the cursor to be set on the related NC block.

Conditional jumps
BES<Label><Event>
BER<Label><Event>

In the conditional jumps BES and BER , the jump to the specified label is always performed in the same way and without evaluating the condition. The cursor can again be positioned within the program. Like in processing, the label is first searched for in the actual program, then in the program 99.

Before the BES or BER jump to the label is performed via the <Plane+> key, you may modify the event via a displayed window (e.g. to influence the synchronization with another process), and jump to the specified label irrespectively of the event's state.

Jumps to other programs

JMP<Progr. No.>

JMP<Variable>

If the cursor is set to a block that contains a JMP command to another program, pressing <Plane+> displays the program that is specified there, and the search for the target and/or start block can be continued there.

With the JMP command, the program number can be transferred using a variable (e.g. JMP @2). When you press the <Plane+> key, the variable value is displayed in a window. At the same time, a new value can be entered for the variable, or a jump can be made to this program, or the action can be aborted.

The search for the target or start block can be continued in the new program.

Subroutine calls

BSR <Label>

JSR<Progr. No.>

BEV<Label><Event>
JEV<Progr. No.><Event>

If the cursor is located on an NC block that contains a subroutine call, pressing the <Plane+> key in conjunction with the commands BSR and BEV causes the specified label to be searched for in the same program or in program no. 99, and the cursor to be set in the located block.

Note: Target and block selection is not possible within a cycle.

With the commands JSR and JEV, the specified program is located and displayed. The cursor sits on the first block N0000 of that program.

Prior to pressing the <Plane+> key for executing BEV or JEV subroutine call, you may modify the event as required. If, in JSR, the program number is transferred via a variable (e.g. JSR @10), a window showing the variable value is displayed prior to the subroutine call. At the same time, a new value can directly be entered for the variable, or the subroutine can be invoked, or the action can be aborted.

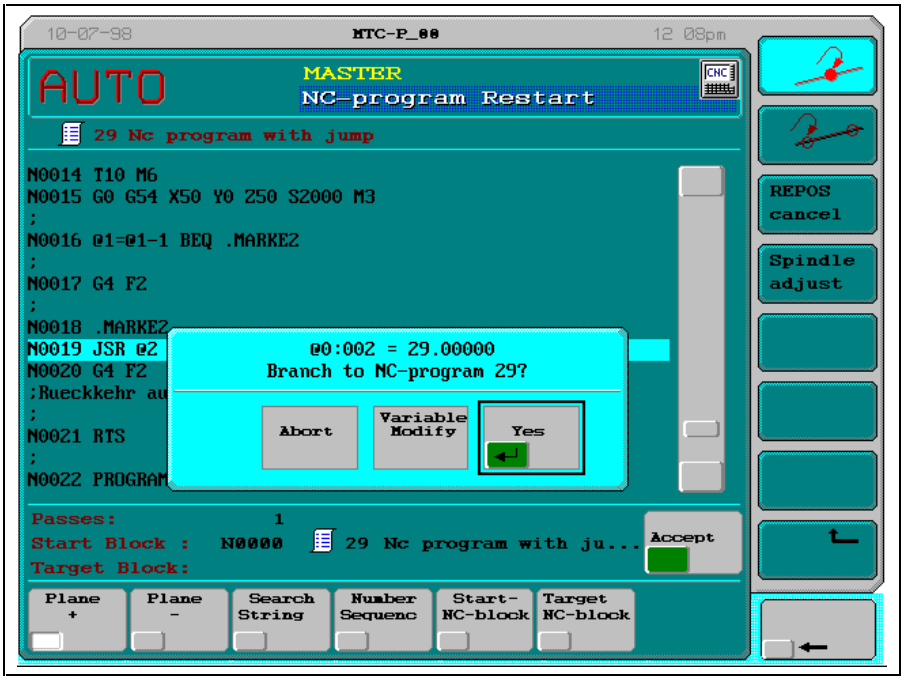

Figure 2-2: Example: Influencing an event with conditional jumps

Return from the subroutine

If you have pressed the <Plane+> to invoke a subroutine and could not locate the required target or start block in the subroutine, for example, you can press the <Plane-> to return from the subroutine to the calling point in the calling program.

Number of subroutine calls

From the main program, any target or start block of the related program(s) and subroutines can be selected using the NC program restart screen (provided that the corresponding block is not in a cycle).If necessary, target or start block can be on the 10th level (9 subroutine calls).The command (BSR, JSR, BEV or JEV) that is used for the call is irrelevant.

2.6 Number of Runs

Number of runs

If the target block is inside a loop or inside several nested loops, you can employ the 'number of runs' function to define the required target block. If you enter a '3' for example, the NC will abort the computing process at the beginning of the target block when it reaches that block for the third time.

Input

Press the <Number of runs> key if you wish to enter the number of runs. Press the <Enter> key to terminate the input.

Value range

1 through 65536

Default value

Upon each entry, the user interface always stores the value '1' for the number of runs (irrespective of any previous entry).

Note: The ´number of runs´ must only be specified if the target block is inside a loop.

After each executed computing process, the NC resets the ´number of runs´ register to '1´.

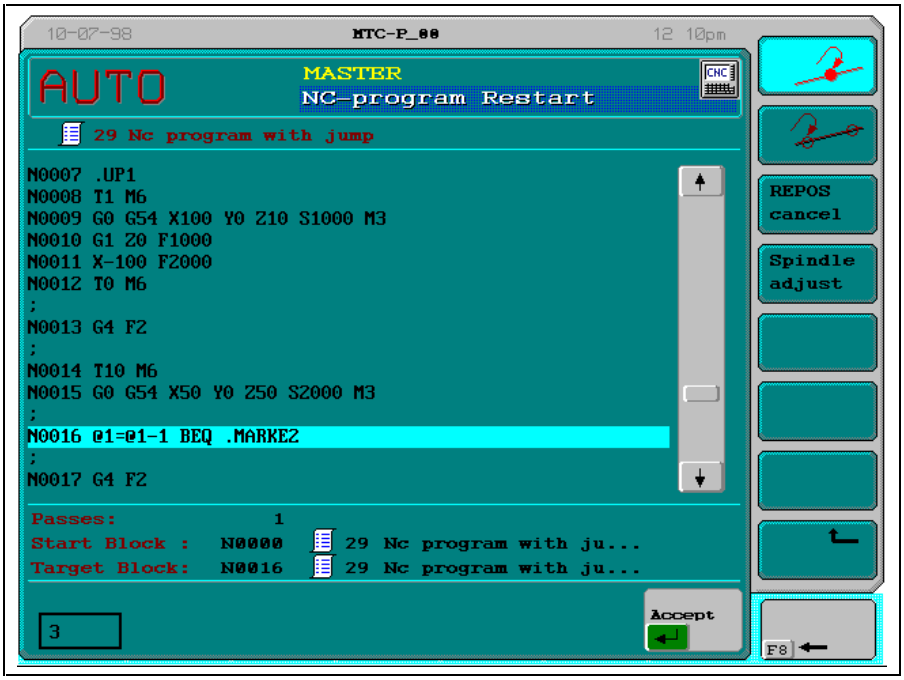

Figure 2-3: Example: Selecting the number of runs

2.7 Starting the Computing Process

Selection

The computing process is usually started automatically when the selection window is closed (see examples). Depending on the configuration, the NC program restart can alternatively be started using a separate M key.

Single-block mode

Selecting single block operation is (like in normal program operation) possible at any time during the computing process.

Computing process

During the computing process, the NC processes the selected NC program like during the normal program mode, taking start and target block into account. However, the controller suppresses the axis movements and the output of the auxiliary functions.

Screen

During computing process, the user interface shows the previously active base screen. Selecting a different station or a different menu is possible at any time.

Target block cannot be found

If the NC encounters an end-of-program command (BST, RET, M02, M30) before it reaches the target block, it issues the error message 'target block cannot be found' and stops program execution.

Note: During the computing process, the stop and the control-reset key have the same functionality as in the normal program mode.

The reverse key is without function during the computing process.

2.8 Establishing the Necessary States

Control states

During the computing process, the NC determines the control states that are required for the program entry. This includes the G functions, the auxiliary functions, and the data that is stored in the machine data of the 'NC program restart and REPOS' page.

Machine states

The NC executes a subroutine called ADJUST to provide the states at the machine that are necessary for the program entry. In this subroutine, the machine manufacturer chiefly provides for outputting the necessary auxiliary functions and for the necessary tool and workpiece change.

Selection

Press the Start key to start the ADJUST subroutine.

Prerequisites

The 'multi-block retrace' function must be activated via the SPS before the ADJUST subroutine is started.

CAUTION

Risk of interference

⇒ The machine must be in a state that permits an automatic execution of the ADJUST subroutine. This means, for example, that the axes are in a safe initial position at which tools and workpieces could be changed without any risk if this were necessary.

Recommendation

At the end of the ADJUST subroutine, an auxiliary function should be used for selecting manual mode. This enables the operator to move the axes to the contour without any interference (like after a program interruption with subsequent jogging).

2.9 Multi-Block Retrace to the Contour

Multi-block retrace

The ´multi-block retrace´ function automates the process of positioning the tool back to the contour after a program interruption or after NC program restart
(see ´multi-block retrace and return to contour´ Description of Functions).

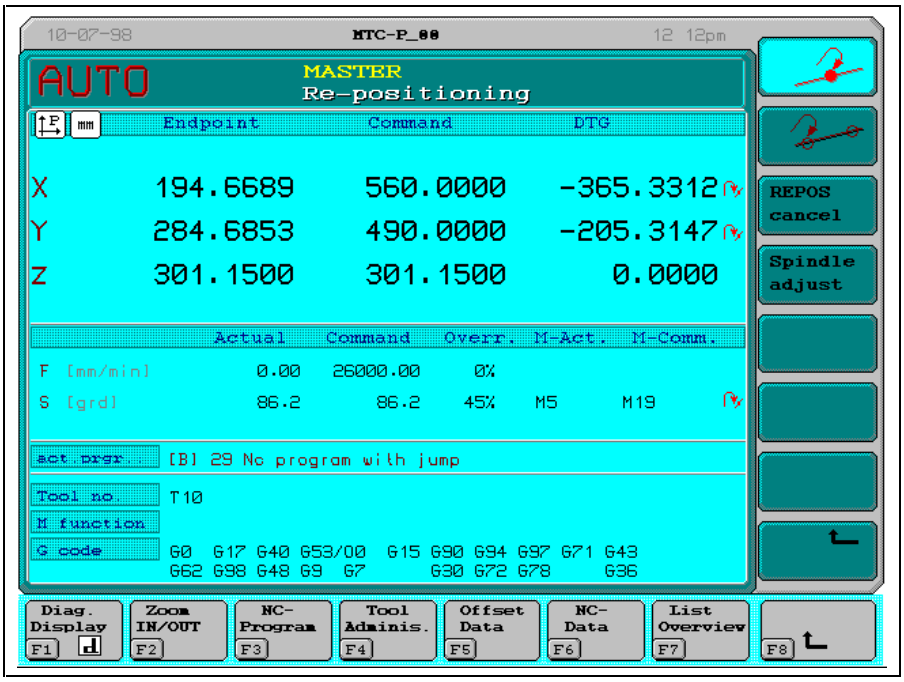

Figure 2-4: 'Re-positioning' screen

Necessary operator input

At the end of the ADJUST subroutine, there is an automatic changeover to manual mode. Like after a program interruption (with subsequent jogging), this enables the user to move the axes to the contour without interference, first in jog mode, and afterwards in automatic mode.

Note: After the ADJUST subroutine has been completed, the NC automatically continues the execution of the REPOS subroutine (provided that manual mode has not been selected before using an auxiliary function).

Selecting manual mode at the end of the ADJUST subroutine enables the operator to move the axes to the contour without any interference (like after a program interruption with subsequent jogging). This is particularly necessary during inside machining and for back cutting.

In this process, the user can jog the axes in manual mode in the vicinity of the re-entry point, then select automatic mode as usual, and continue automatic multi-block retrace and the further program execution without any further interruption.

---

3 Machine Parameters

3.1 Auxiliary Function Output During NC Program Restart

Designation

- M function output during NC program restart
- Q function output during NC program restart
- S function output during NC program restart
- T/E function output during NC program restart

Number Bxx.058 - Bxx.061

Value range Yes / No

Default value No

Unit -

From version onwards 18V00

Dependency -

Purpose The process parameter ‘M, Q, S and T/E function output during NC program restart’ define the output behavior of the auxiliary function concerned during mid-program startup.

Explanation If the machine manufacturer sets ‘output behavior upon NC program restart’ to ‘No’, the NC will not issue the auxiliary function concerned to the SPS.

If, in contrast, the parameter is set to ‘Yes’, the NC transfers the auxiliary functions concerned according to the selected output and acknowledgment behavior to the SPS, as it does during the normal operation. During mid-program startup, the machine manufacturer can collect the auxiliary functions in the SPS, interpret them and output them upon the next program start.

Irrespective of the selected output behavior, the auxiliary functions that establish the states at the machine that are necessary for the entry into the NC program, must be output at the end of the execution. This is done using the ‘.ADJUST’ subroutine.

Peculiarities *1. M function groups M(1) through M(15)*

Irrespective of the ‘output behavior upon NC program restart’, the NC collects the auxiliary functions for each auxiliary function group during mid-program startup (like during normal program operation), and saves the auxiliary function last output of each of the groups (M(1) through M(15)).

With the exception of the spindle and gear groups, the groups have the default value -1 assigned (after the controller has been switched on and at the end of the program (BST, RET, M(30)). Use M05, M105, M205 or M305 to assign the default values to the spindle groups.

Note: In manual mode, the auxiliary spindle functions are updated when the spindles are jogged. The controller does not update any other auxiliary function (such as 'coolant ON/OFF') that are output during manual mode.

2. M function group M(16) and Q functions

For the M function group 16 and for the Q function group, the NC saves the last 40 auxiliary functions in the machine data of the 'M(16) and Q function buffer' page. This machine data has '0' preassigned.

Within the M(16) group, the 'M(16)' address can be used for reading the auxiliary function last output. The same is true for the Q functions. The M(16) and Q function groups have the default value -1 assigned (after the controller has been switched on and at the end of the program (BST, RET, M(30)).

Note: The NC does not output the program control commands M00, M01, M02 and M30 to the SPS like all the other M functions if the machine manufacturer sets the 'M function output during NC program restart' parameter to 'No'.

4 Machine Data

4.1 Machine Data Page ‘Block Search and REPOS’

Designation NC program restart and REPOS

Page No. 60

From version onwards 18V00

Purpose The ‘NC program restart and REPOS’ page defines the spindle mode.

Explanation For each process, the ‘NC program restart and REPOS’ page is of the following structure:

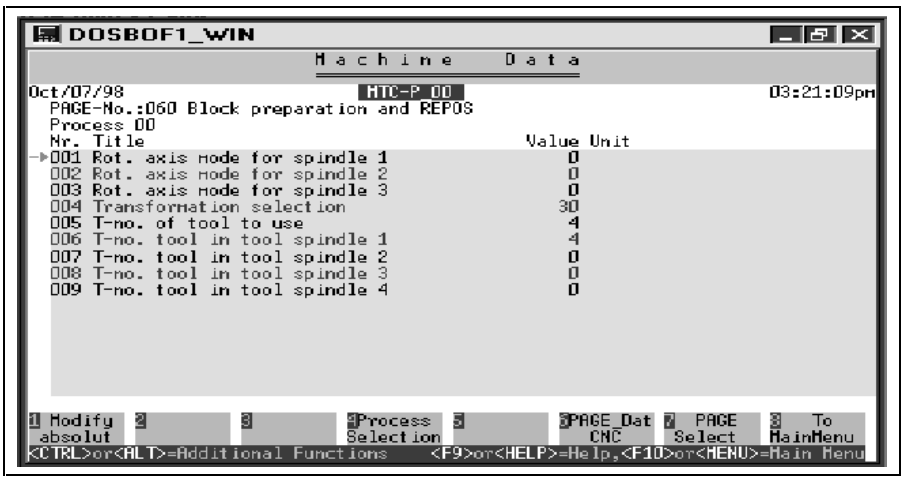

Figure 4-5: Data elements per process of the ‘NC program restart and REPOS’ page

The following table shows the value ranges, default values, and updating time of the data in the ’NC program restart and REPOS’ page.

Data element	Value range	Default value	Updating time
Rotary axis mode for spindle 1 / 2 / 3	0: Spindle mode must be established for the 1st/2nd/3rd spindle of the process. 1: Rotary axis mode must be established for the 1st/2nd/3rd spindle of the process.	0	upon program stop and at the end of the block search when the target block is reached.
Transformation selection	30: Transformation need not be selected 31: G31 must be selected 32: G32 must be selected	30	at the end of the block search when the target block is reached.
T no. of the tool in tool spindle 1 / 2 / 3 / 4	0 through 9999999	0	at the end of the block search when the target block is reached.

Note: The states of the physically existing spindles that are defined in the 'rotary axis mode for spindle 1 / 2 / 3' data elements of the 'NC program restart and REPOS' page must be established within the '.REPOS' subroutine.

The transformation for facing (G31) or lateral cylinder surface machining (G32) must be selected according to the 'transformation selection' machine data element on the ´NC program restart and REPOS´ page within the ´. ADJUST´ subroutine.

If the individual processes contain at least one tool spindle, this tool spindle (and any additional tool spindles) must, within the ´.ADJUST´ subroutine, be equipped with the tools that are specified in the ´T no. of the tool spindles 1 / 2 / 3 / 4´ data elements of the ´NC program restart and REPOS´ page.

4.2 M(16) and Q Function Buffers

Designation M(16) and Q function buffers

Page no. 61

From version onwards 18V00

Purpose The last 40 M functions that were output within the M function group 16 during the NC program restart, and the last 40 Q functions that were output during the NC program restart can be read at the end of the computing process, and be output to the SPS.

Explanation The ´M(16) and Q function buffer´ page is of the following structure:

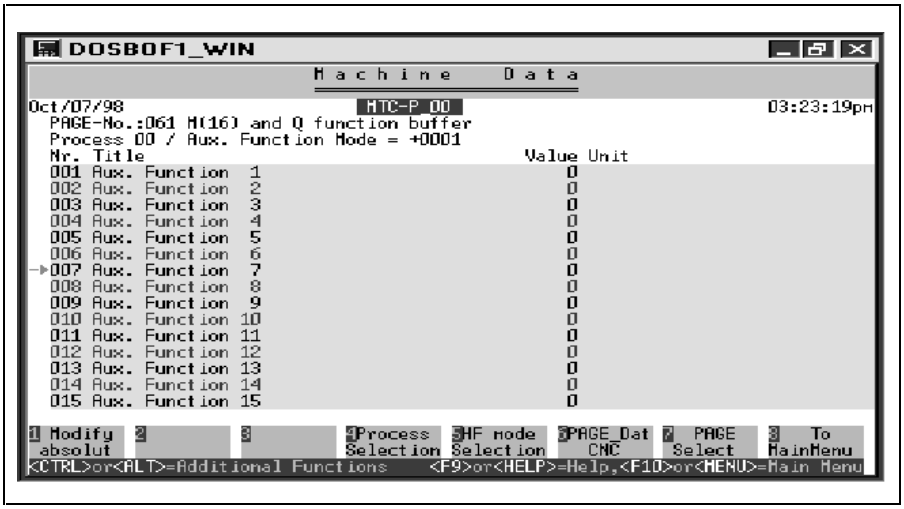

Figure 4-6: Data elements per process and M/Q function group of the ´M(16) and Q function buffer´ page

During NC program restart, the NC stores the last 40 M function numbers that were output within the M function group 16. Likewise, it stores the last 40 Q function numbers that were output during NC program restart.

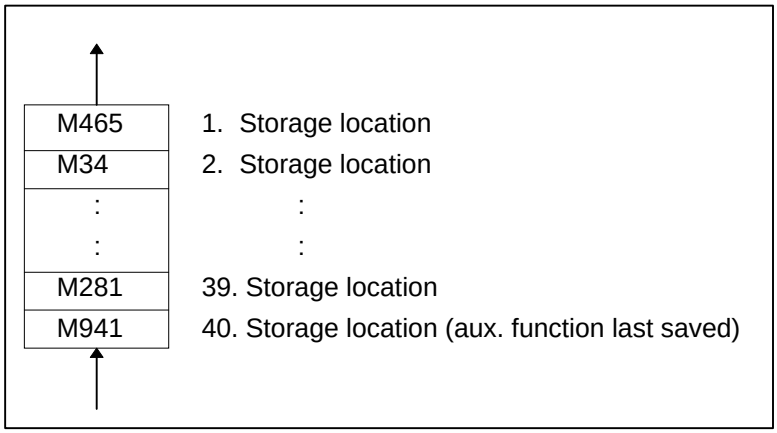

Figure 4-7: Method of operation of the auxiliary function buffer

Note: The NC deletes the auxiliary function that was stored first, and stores the subsequent auxiliary functions in the next lower memory location within the ´M(16) and Q function buffer´ machine data page as soon as the auxiliary function buffer overflows (i.e. the 41st M function of group 16 or the 41st Q function must be stored during the computing process). Thus, the new auxiliary function is stored as the 40th auxiliary function.

M19 commands (M19, M119, M219 or M319) are not stored within the ´M(16) and Q function buffer´ page.

Default value At the beginning of the NC program restart, the NC initializes all data elements of the ´M(16) and Q function buffer´ page that belong to the process with the value '0'.

5 NC Syntax

5.1 Primary Block Programming

Primary block

66257 To DIN 66257, a primary block is a block that contains all the words (NC commands) that are required for (re)starting the machining sequence. A primary block starts with the primary block character.

Primary block character

DIN 66025 (part 1) defines a colon ´**:**´ as the primary block character. Within an NC block, the colon must be programmed as the first character.

Note: If a primary block shall also be a skipped block, the skip slash ´/´ must be programmed before the colon.

A primary block can also be programmed in a subroutine.

When primary blocks are programmed, it must be ensured that the start block lies before an incremental dimension (G91) and before an incremental zero offset (G51).

Application

A primary block can be used

- as a target block in block search (without computation) or
- as a start block in NC program restart (with computation).

Usually, primary blocks are used for NC program restart (in particular if tools or workpieces are supplied during the program execution from a different process or from the outside).

Example

In order to ensure that the operator has a swift program entry during block search and/or NC program restart, primary blocks were programmed in the following programming example.

```
        :
; stair II turning
N0825 : T12 M6
N0826  G92 S2800
N0827  G0 G6 G8 G54 G97 X50 Z20 S1500 M3
N0828  G1 X48 F2000
        :
```

5.2 Auxiliary Function Buffer

Auxiliary function buffer

The auxiliary functions last output during NC program restart within the individual M, Q, S, T or E function groups can be read at the end of the computing process, and be output to the SPS.

Note: At the end of program and upon Control-Reset, the NC initializes all memory locations of the M and Q function groups with the value ´-1´. The spindle and gearstep groups are the sole exception.

Besides the M, Q, S, T and E function group, the NC stores the last 40 M functions of the M function group 16 and the last 40 Q functions during the NC program restart within the ´M(16) and Q function buffer´ page.

Note: M19 commands (M19, M119, M219 or M319) are not stored within the ´M(16) and Q function buffer´ page.

5.3 ADJUST Subroutine

The ADJUST subroutine enables the machine manufacturer to establish the states that are necessary for the program entry at the machine. In this subroutine, the machine manufacturer chiefly provides for outputting the necessary auxiliary functions and for the necessary tool and workpiece change.

The ´.ADJUST´ jump label must be set at the beginning of the subroutine. Furthermore, the ADJUST subroutine must be written before the REPOS subroutine (without RTS !). This enables the NC to automatically continue with the execution of the REPOS subroutine once it has completed the execution of the ADJUST subroutine.

If a reverse occurs while the ADJUST subroutine is being executed, the NC jumps to the related label and continues execution there before it jumps to the ´.HOME´ label.

The user can store the ADJUST subroutine in the current NC program, in the program 99, or in the cycle memory.

Example Within the ADJUST subroutine, the necessary states (that are specified through auxiliary functions) at the machine are set for a subsequent multi-block retrace.

```
;-------------------------------------------------------------------------------------------------------------
;   Subroutine:  ADJUST
;-------------------------------------------------------------------------------------------------------------
;
; Setting the M functions of the M(5) through M(15) group
; Group 5
.ADJUST @100=M(5)-1  BMI .INI_G5        ;Initialization value ?
M=M(5)  BRA .ADJ_G6                     ;Updating M function group 5
.INI_G5  M109                           ;Coolant OFF
; Group 6
.ADJ_G6  @100=M(6)-1  BMI .INI_G6       ;Initialization value ?
M=M(6)  BRA .ADJ_G7                     ;Updating M function group 6
.INI_G6  M209                           ;Coolant OFF
              :
              :
; Group 15
.ADJG15  @100=M(15)-1  BMI .INIG15      ;Initialization value ?
M=M(15)  BRA .ADJUS1                    ;Updating M function group 15
.INIG15  M918                           ;Chip conveyor OFF
;
; Updating M functions of the group 16
.ADJUS1  @100=1                         ;Run index
.ADJ_12  @101=MTD(61, 0, 1, @100)-0     ;Page:61, process:0, auxiliary function:1(M functions)
BEQ .ADJUS2                             ;Value=0 ?
@101=MTD(61, 0, 1, @100)-500 BEQ .ADJ_13        ;Value=500 ?
@101=MTD(61, 0, 1, @100)-501 BEQ .ADJ_13        ;Value=501 ?
M=@101                                  ;Auxiliary function output
.ADJ_13 @100=@100+1                     ;Increasing run index
@102=40-@100  BPL .ADJ_12               ;Value>=40 ?
;
; Updating Q functions
.ADJUS2  @100=1                         ;Run index
.ADJ_22  @101=MTD(61, 0, 2, @100)-0     ;Page:61, process:0, auxiliary function:2(Q functions)
BEQ .ADJUS3                             ;Value=0 ?
@101=MTD(61, 0, 2, @100)-5600  BEQ .ADJ_23      ;Value=5600?
@101=MTD(61, 0, 2, @100)-5700  BEQ .ADJ_23      ;Value=5700?
M=@101                                  ;Auxiliary function output
.ADJ_23 @100=@100+1                     ;Increasing run index
@102=40-@100  BPL .ADJ_22               ;Value>=40 ?
;
; Tool change
.ADJUS3
 T=T  BSR .M6                           ;Changing in tool last selected
;
;-------------------------------------------------------------------------------------------------------------
;   Subroutine:  REPOS
;-------------------------------------------------------------------------------------------------------------
              :
              :
```

5.4 Peculiarities

NC-Specific Peculiarities

Basic method of operation

With a few exceptions, the NC processes the NC commands during NC program restart like during automatic mode. The exceptions are listed below.

Dwell

The NC does not take dwells (G04) into account during the computing process.

Transformation for facing / lateral cylinder surface machining

The transformation for facing (G31) or lateral cylinder surface machining (G32) must be selected at the end of NC program restart according to the 'transformation selection' machine data element on the ´NC program restart and REPOS´ page within the ´. ADJUST´ subroutine.

Homing axes

The drives do not perform the 'Homing axes´ (G74) function during NC program restart. Thus, the actual position does not change while a G74 command is being processed.

Driving against hard stop

When the NC executes the 'driving against hard stop' command (G75) during the computing process, it does not move the drive concerned. The motor current is not increased due to a mechanical resistance.

Spindle control commands

With NC-controlled spindles, the NC does not execute the spindle control commands (Mj03, Mj04, Mj05, Mj19; with ´j´ = 1, 2, or 3) during the computing process. It merely issues the auxiliary functions to the SPS like in normal operation if this has been preselected in the process parameters.

The NC retains the states of the spindles with rotary axis capability that are necessary for the subsequent program entry and saves them in the ´NC program restart and REPOS´ page. The states that are necessary for program entry have been produced depending on the required modes (like after an interruption) using G77 Sj=0 (with j = ´ ´, ´1´, ´2´, ´3´) or G77 C0 (C: axis designation of the rotary axis). The mode that must be established can be taken from the machine data page 60 ´NC program restart and REPOS´.

Example:

For a spindle with rotary axis capability and the designation S1/C1, the mode (spindle / rotary axis mode) that is required for the further operation must be established in the REPOS subroutine after an interruption and/or the NC program restart.

```
.REPOS
          :
@100=MTD(60, 0, 0,1)-1 BEQ .ADJ_5   ;Page:60, process:0, , data element: 1
                                     (rotary axis mode for S1)
G77  S1=0                           ;Spindle mode selection
G77 C1=0                            ;Rotary axis mode selection
          :
```

Program control commands and gear changing functions

The NC executes the program control commands (M00, M01, M02, M30) and the gear changing functions (Mj40, Mj41, Mj42, Mj43, Mj44; with j = 1, 2, or 3) without any restrictions, like during normal program operation. The NC transfers them to the SPS, if the machine manufacturer has preselected it in the process parameters.

Process control commands and process synchronization commands

during NC program restart, the NC executes the process control commands (DP, SP, RP, AP, WP, LP and POK) and the process synchronization commands (WES and WER) like in normal operation. A prerequisite is, however, that the individual programs have been started in the individual processes like in normal operation.

Recommendation

If the other processes shall not be taken into consideration (i.e. they shall not be influenced) during the NC program restart, because they already have reached the required program locations in normal operation or via NC program restart, this can be achieved using a skip slash or a conditional jump instruction before the process control commands or process synchronization commands.

Data exchange with digital drives that are equipped with SERCOS interfaces

During the computing process with digital drives (with SERCOS interface), data exchange using the AXD command is performed like in normal program operation. This is also true for APR-SERCOS parameters that are processed by the APR.

Note: An auxiliary function, which the NC does not output during NC program restart and/or the SPS does not process during NC program restart, must be used via the SPS for activating AXD commands that cause a movement.

Example:

Certain functions (such as AXD commands) should not be output to the controller during NC program restart.

There are different implementation possibilities for this purpose:

1. Conditional jumps

Using an event the SPS sets during the NC program restart permits specific functions to be skipped that shall not be executed in the NC during NC program restart:

```
      :
BES .BP1 1:15
AXD (X:S-0-0104)=7000      ;new KV factor for X axis
AXD (Y:S-0-0104)=7000      ;new KV factor for Y axis
AXD (Z:S-0-0104)=7000      ;new KV factor for Z axis
.BP1
      :
```

2. Implementation in the SPS

In this solution, the SPS executes the functions that must not be processed during NC program restart (such as the AXD commands). As long as these functions are activated using an auxiliary function type (e.g. using Q functions), their output to the SPS during NC program restart can easily be implemented. It merely requires the related process parameter

Bxx.057 Q function output during mid-program startup Yes/No

to be set to No. The associated NC program could then look like this:

```
      :
Q101      ;new KV factor for X axis
Q102      ;new KV factor for Y axis
Q103      ;new KV factor for Z axis
      :
```

Axis transfer

The NC processes the axis transfer commands (GAX and FAX) like in normal program operation. If the NC encounters an FAX command during the computing process, it waits until the axis is requested by another process (that may also be in the NC program restart) using GAX. Accordingly, the NC waits upon an axis request using GAX until another process (that may also be in the NC program restart) relinquishes the axis concerned using FAX.

Reading the position value

The NC does not interpret the PMP and NMP commands (that are used for acquiring the current actual position of analog drives) during the computing process.

Tool-Management-Specific Peculiarities

Tool-specific inquiries

The NC processes the conditional jump instructions ´BSE´ (jump if tool spindle is empty) and ´BTE´ (jump if T0 has been programmed) like during normal program operation. During NC program restart, however, it refers to the (virtual) tool spindles that are mapped within the machine data rather than to the real tool spindles.

In contrast, the NC does not take the commands for checking spindle 1 ´TSE´ and a tool magazine location in position 1 ´TPE´ into account during the computing process.

Current tools list

Besides the correction data of a process, the current tools list contains all data items that are required for the management of all tools that belong to the process. During NC program restart, the tool management does not change the data of the current tools list.

Automatic equipment check

The NC performs the automatic equipment check in the same way as it does during the normal program operation when the process-related interface signal ´PxxCMGNTC´ is not set.

Mechanisms for providing tools

The following table shows the different mechanisms that are used for loading, changing in and activating a tool as a function of the tool storage unit.

Mechanism	Type	Turret	Magazine
Load and activate tool	synchronous and process-internal	Txxx MTP([POS], [DIR]) Txxx MMP([POS], [DIR]) (Parameter ´asynchronous turret movement´ = NO)	
	asynchronous and process-internal	Txxx MTP([POS], [DIR]) Txxx MMP([POS], [DIR]) (Parameter ´asynchronous turret movement´ = YES)	
Load tool	asynchronous and process-internal	-	Txxx MTP([POS], [DIR]) Txxx MMP([POS], [DIR]
Changing tool into tool spindle and activating it	synchronous and process-internal	-	TCH([POS], [SPI]) TMS([POS], [SPI])
Changing tool into tool gripper	synchronous and process-internal	-	-

Abbreviations:

[ ]:	optional parameter
DIR:	direction
POS:	position 1, 2, 3 or 4 to be approached
SPI:	tool spindle 1, 2, 3 or 4

Note: Process-overlapping tool changing processes (that are not contained in the table above) are not supported during NC program restart. This means for these applications that the tool that is required for machining must be changed into the corresponding spindle and the NC program restart must be started from the last primary block so that a process-overlapping tool change is not necessary during NC program restart.

Tool storage unit motion commands for loading and activating a tool within a process

The tool storage motion commands MFP, MHP, MMP, MOP, MRF and MTP are used for triggering the necessary movement of the tool storage unit. The MRY command is used for interrogating whether the movement of the tool storage unit is finished.

If the tool storage unit is a turret (process parameter ´type of tool storage´ = turret), the MTP and MMP commands are used either

- for synchronous loading and activation of the tools (process parameter ´asynchronous turret movement´ = NO) or
- for asynchronous loading and activation of the tools (process parameter ´asynchronous turret movement´ = YES)

With a magazine (process parameter ´type of tool storage´ = magazine), in contrast, the tool storage motion commands MTP and MMP are used exclusively for asynchronous and process-internal tool loading (motion control of the magazine).

To avoid illegal movements, the NC does not perform the MFP, MHP, MMP, MOP, MRF and MTP motion commands and the MRY interrogation. With SPS-controlled tool storage units, this means that during the computing process the NC does not send the SPS the position that shall be moved to and does not activate the MMV and MRF standard functions.

Loading tools from a different process

During NC program restart and within a process, the NC can only access the tool data that is available within the process. Thus, NC program restart is not possible through machining sequences that employ tools that are outside the process concerned. Th NC generates a corresponding error message in such a situation.

Synchronous and process-internal tool transfer commands

The synchronous tool transfer commands

- TCH (complete tool change between magazine and spindle),
- TMS (tool change between magazine and spindle), or
- TSM (tool change between spindle and magazine)

are used for changing the tools between the tool spindles or tool grippers that belong to a process and the tool storage unit. They permit a (logical and physical) tool change within a process. Machining is not possible as long as they are handled at NC and SPS side (performing a tool change).

During the computing process, the NC does not report the synchronous tool transfer commands to the SPS and does not activate and/or deactivate the tool concerned. The current tools list remains unchanged. Thus, it does not return the tool in the specified spindle back into the tool magazine and does not take the tool from the tool changing position to the specified spindle. However, it remembers the tools that shall be changed in and the spindles into which the tools are to be changed in. Thus it is able to access the correction data of the tool in the current tool spindle in order to calculate the path.

Asynchronous tool transfer commands

Using the standard functions

- XFER_CHK, XMS, XSM, XMG, XGM, XSG, XGS or
- TL_MOVE and/or
- TLBD_RD, TLED_RD, TLBD_WR, TLED_WR, TL_ENABL,

the SPS can change tools asynchronously to machining and in a process-overlapping fashion.

Asynchronous tool change initiated from the machining program

Asynchronous and process-overlapping tool changes into the tool spindles and tool grippers are usually initiated via the machining program and executed in parallel to the machining process in a cooperation between the SP and an auxiliary process (that usually includes background magazine).

Since tool data is missing, an asynchronous tool change that is, for example, initiated via M functions (tool change M functions) in the machining program and is not performed during the NC program restart will result in an error message being issued.

Asynchronous tool change initiated from the SPS

Transferring tools in and out and resorting tools is usually controlled by a separate process (background process), the SPS or an external system, not by the machining process.

If, for example, the SPS employs the asynchronous tool changing function to perform a tool change during NC program restart, it modifies the current tools list with this process. Thus, asynchronously loading or unloading a tool storage unit and resorting tools during NC program restart is done like during normal program operation.

T word

During operation, the NC stores the T number last programmed in the T word. Depending on the motion commands MTP and MMP, the NC interprets the T number as a tool number or as a tool location number.

During NC program restart, the NC stores the T number under the T address, like in normal operation. It can be read and re-specified in the ´.ADJUST´ subroutine.

Note: The NC generates an error message if the tool that is selected via the T number is not contained in the current tools list during the computing process.

---

6 Interface Signals

6.1 NC Program Restart is Active

Designation PxxSBPACT (NC block preprocessing is active)

Task The process-related status signal ´NC program restart is active´ indicates to the SPS whether or not the NC program restart function is active.

Interpretation The NC sets this signal only after all the following conditions have been satisfied:

- The 'ready to start' signal (PxxSREADY=1) is applied for the process.
- The controller contains the program that shall be executed.
- The program that shall be executed is selected.
- Automatic mode (or semi-automatic mode or program execution in setup mode) is selected.
- No program is active (PxxSACTIV=0)
- The operator has selected NC program restart and properly selected the target block.

The NC resets the signal after NC program restart or the related computing process has been performed and/or after NC program restart has been aborted using Stop and Control Reset.

Meaning NC program restart is active as long as the ´NC program restart is active´ signal is ´1´. NC program restart is no longer active when the signal assumes the value ´0´:

PxxSBPACT=0: NC program restart is not active

PxxSBPACT=1: NC program restart is active

6.2 Starting NC Program Restart (Computing Process)

Designation PxxCBPSTR (NC block preprocessing start)

Task Setting the process-related control signal ´start of NC program restart´ starts the NC program restart or, rather, the computing process proper from the SPS.

Interpretation The NC only interprets the ´start of NC program restart´ if the prerequisites for NC program restart have been fulfilled and if this is announced via the ´NC program restart is active´ signal (see above).

Meaning The NC sets the ´start NC program restart´ (positive edge) signal to start NC program restart (provided that the prerequisites for NC program restart have been fulfilled):

PxxSBPACT $0 \rightarrow 1$: Activating NC program restart

6.3 Interaction of the Interface Signals

Overview

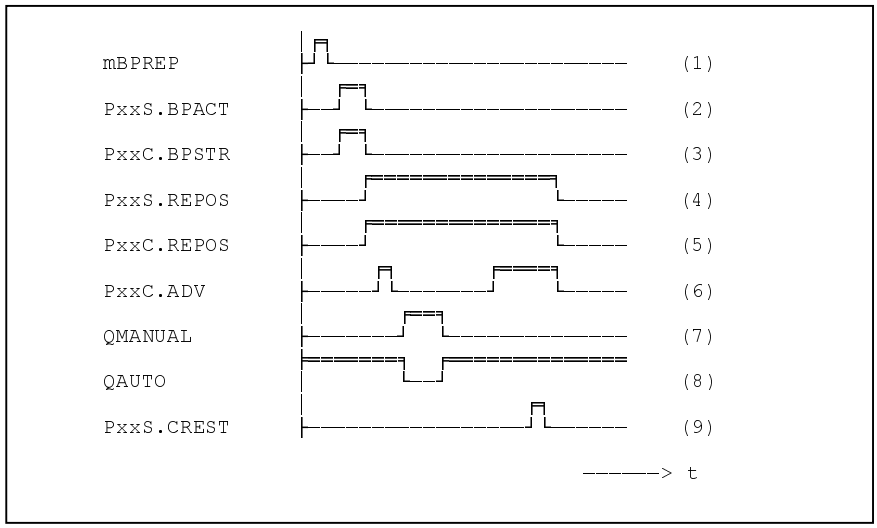

Figure 6-8: Interaction of the relevant signals for NC program restart and subsequent multi-block retrace

Displaying the NC program restart screen

(1) The user interface displays the NC program restart screen when the M key ´NC program restart´ (mBPREP) is pressed.

NC program restart is active

(2) Provided that the operator has properly selected the start block and the target block, the controller sets the ´NC program restart is active´ (PxxSBPACT), status signal when the screen is exited.

Starting NC program restart (computing process)

(3) Once NC program restart is active, the SPS automatically sets the process-related ´start NC program restart´ (PxxCBPSTR) control signal to start the computing process proper.

NC program restart terminated

(4) Once the target block has been reached, the NC resets the ´NC program restart is active´ (PxxSBPACT) signal. At the same time, it sets the ‘multi-block retrace/return to contour possible’ (PxxC.REPOS) interface signal.

Activating multi-block retrace

(5) As soon as the NC reports ´multi-block retrace/return to contour possible´, the SPS sets the ´multi-block retrace´ (PxxCREPOS) interface signal to activate repositioning to the contour.

Starting the ADJUST subroutine

(6) When the Start key (PxxCADV) is pressed, the NC starts with executing the ADJUST subroutine.

Changeover to manual mode

(7) At the end of the ADJUST subroutine, manual mode (QMANUAL) is selected using an auxiliary function.

Changeover to automatic mode

(8) Reselect automatic mode (QAUTO) after you have adjusted the axes such that interference is not possible even in exceptions (such as inside machining or back cutting)

Starting the REPOS subroutine

(6) When the start key (PxxCADV) is pressed, the NC starts processing the REPOS subroutine, establishing the states that are required for the subsequent machining (starting the NC program at the selected target block).

Required state established

(9) As soon as all axes have been adjusted, the NC sets the ’state before interruption re-established’ interface signal to report this state, and automatically continues program execution.

6.4 Behavior of the Interface Signals

Basics

With a few exceptions, the NC processes the interface signals in the same fashion as it does in automatic mode. The exceptions are described in the following sections.

Axis Control Signals

Axis ready for operation

Albeit the NC does not take the ´axis ready´ (AxxCREADY) axis control signal into consideration during the computing process, it transfers it, combined with further conditions, to the SPS as the 'controller enabling' status signal (AxxSRF), like in normal program mode.

Position strobe

The NC does not process the PMP and NMP commands during NC program restart. Thus, the ´position strobe´ (AxxCSTRBP) signal will not be considered.

Motion hold

The NC ignores the motion hold (AxxCMHOLD) signal during the computing process.

Changing gears

The gear step bits (AxxCGEAR1, AxxCGEAR2, AxxCGEAR3) that are sent from the SPS remain unconsidered during the computing process. This is also true if a multi-step gearbox must be switched during the computing process. In this process, the NC performs the programmed gear shift only logically. During the computing process and up to the next gear shift (Mj40 through Mj44; with j = 1, 2 or 3) it accesses the data of the logically activated gear step.

Spindle command speed reached

The spindle speed is not changed during NC program restart. During the computing process, the NC does not consider the ´spindle command speed reached´ (AxxCN_CMD) axis control signal, which it interprets for external spindles. With the corresponding spindle control commands it does not wait for the ´spindle command speed reached´ signal but continues program interpretation without interruption.

Override

The NC does not consider the override signals (PxxCFOVRD, PxxCROVRD) during the computing process. In contrast, the NC takes the axis-specific override (AxxCOVRD) with spindles and NC-controlled tool storage axes into account when the user moves the NC-controlled tool storage axis during the computing process in manual mode.

Process / Mechanism Control Signals

Minimizing the torsion torque

The NC does not interpret the ´minimize torsion torque´ (PxxCSS1MT, PxxCSS2MT) signals during the computing process.

Override

The NC does not consider the override signals (PxxCFOVRD, PxxCROVRD) during the computing process.

Spindle synchronization

The activation of main spindle synchronization during the computing process must be suppressed using the related interface signals (PxxCSS1ON, PxxCSS2ON). main spindle synchronization must be switched off at the beginning of the computing process.

The synchronization between the spindles involved must be established in the ´.ADJUST´ subroutine.

Tracking

Tracking is switched on/off during the computing process using the interface signals (PxxCSCON1, PxxCSCON2, PxxCSCON3, PxxCSCON4) like in normal operation.

The slave compound must be resolved in the ´.ADJUST´ subroutine before the slave axes are adjusted. Next, the axes must be adjusted. Finally, the slave compound must be re-activated.

Axis Status Signals

Waypoints

Like in normal operation, the NC updates the waypoint signals AxxSWP0 through AxxSWP7 by comparing the individual waypoints with the current actual position. Since, during NC program restart, a change of the actual axis positions can only be caused y external influence, the states of the waypoint signals only change when the axes are forcibly moved

Spindle signals

The NC transfers the spindle-related AxxSLD90, AxxSSYNC, AxxSN_CMD, AxxSN_MIN, AxxSMD_DX, AxxSN_MAX, AxxSINPOS and AxxSP_PX status signals to the SPS like in normal operation. During NC program restart, however, the NC does not move the spindle drive.

Motion announcement

The NC does not process the motion announcement signals (AxxSMCPOS, AxxSMCNEG) during the computing process. At the beginning of the computing process, the NC sets these signals to ´0´ and updates them after NC program restart has been deselected.

In-position window

During NC program restart, the ´in-position window´ (AxxSPOSWN) status signal is related to the last command position that was specified before NC program restart was switched on.

Process / Mechanism Status Signals

Tool life monitoring

During the computing process, the NC modifies the data of the current tools list only during the automatic equipment check. As a result, it modifies the status signals that are used for indicating the tool life count (PxxSMGWRN, PxxSMGTWO) only during the automatic equipment check.

6.5 Standard Functions and Function Blocks

Basics

During NC program restart, the standard functions and standard function blocks show, with a few exceptions, the same behavior as during automatic mode. The exceptions are described below.

Auxiliary Functions:

During the computing process, the standard functions that are used for interrogating the M, Q, S, T and E functions (M_FKT, Q_FKT, S_FKT, T_FKT, E_FKT, M_ALL, Q_ALL, S_ALL, T_ALL, E_ALL, M_NR, Q_NR, S_NR, T_NR, E_NR) do not signal any auxiliary function that is to be processed if the related ´M, Q, S and T/E function output during NC program restart' process parameters were set to NO.

If the machine manufacturer sets the ´M, Q, S and T/E function output during NC program restart' process parameters to YES, the standard functions that are used for interrogating the M, Q, S, T and E functions report the auxiliary functions that are to be processed in the same way as during normal program mode.

Spindle with rotary axis capability

During the computing process, the NC does not update the standard functions that are used for selecting spindle mode or rotary axis mode (SPMOD, ROTMOD). For each spindle, it stores the mode last selected in the 'rotary axis mode for spindle 1/2/3' data elements of the machine data on the 'NC program restart' page during the computing process.

Combined spindle/turret axis

During the computing process, the NC does not update the standard functions that are used for selecting spindle mode or turret mode (MAG_ACT, SPDL_ACT). It does not store the mode either because turret mode only exists briefly as long as the turret is swiveled.

7 Examples

7.1 Simple Milling Machine

Task

NC program restart including multi-block retrace must be commissioned for a simple milling machine (see figure below).

Structure of the milling machine

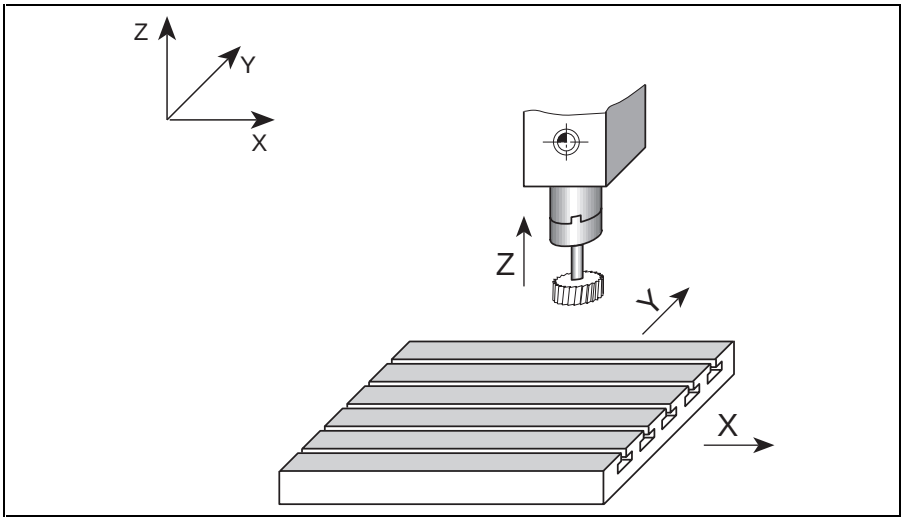

Figure 7-9: Structure of the milling machine

The milling machine mainly consists of a horizontal table that is moved along the two axes X and Y. The vertically arranged Z axis moves the tool perpendicularly to the table. Furthermore, there is a spindle that accommodates the tools. A double gripper fetches the tools from an SPS-controlled tool magazine with 24 locations.

S, T and M functions are employed for the operation of this milling machine. All M functions are assigned to the groups 1 through 15.

Parameter value assignment

Bxx.058	M function output during NC program restart	No
Bxx.059	Q function output during mid-program startup	No
Bxx.061	S function output during mid-program startup	No
Bxx.061	T/E function output during mid-program startup	No

Configuring the M keys

You must select manual mode at the end of the ´.ADJUST´ subroutine. This enables the operator at the beginning of the multi-block retrace process to approach the axes to the workpiece such that interferences can be excluded during the subsequent automatic adjustment.

During manual mode, the operator should be provided with the same M key level as during NC program restart (level 4, see figure below) in automatic mode. Using the global flags, a second manual mode screen is introduced for this purpose. In contrast to the first one, this second screen invokes M key level 4. The second manual mode screen is triggered via the mMANUAL2 auxiliary flag (see the screens that are shown below).

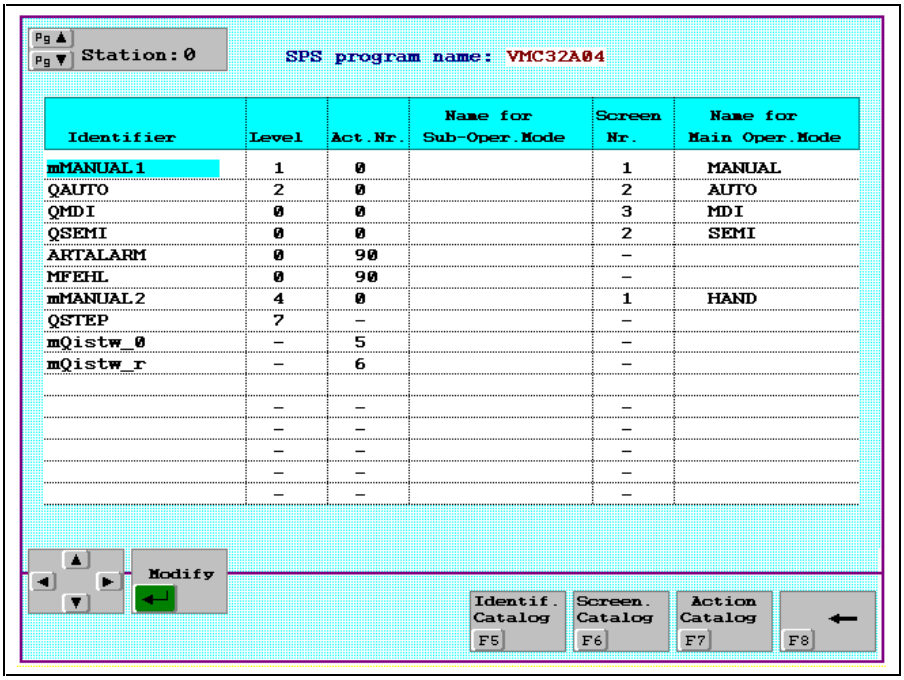

Figure 7-10: Entering the global flags

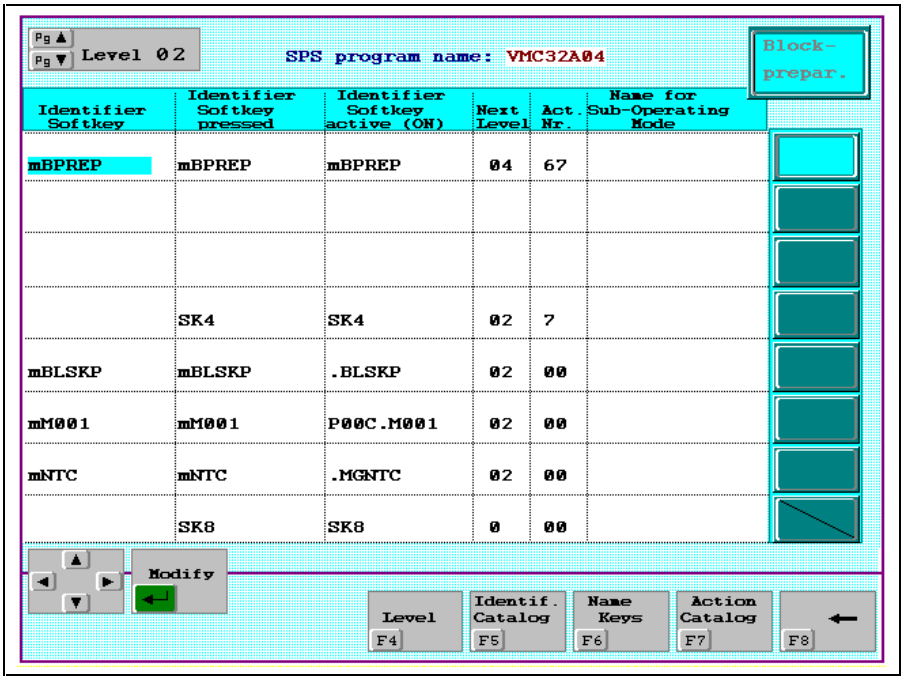

Figure 7-11: Entering the M key ´NC program restart´

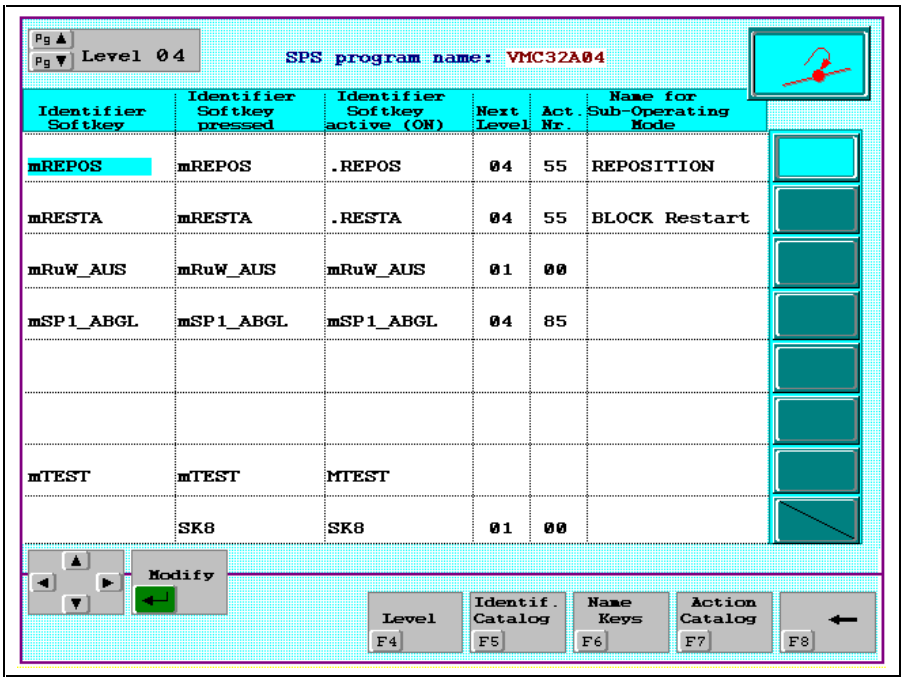

Figure 7-12: Entering the M key ´REPOS´

SPS program segment for NC program restart

The following figure shows the SPS program segment that is used for implementing the NC program restart.

NC program restart

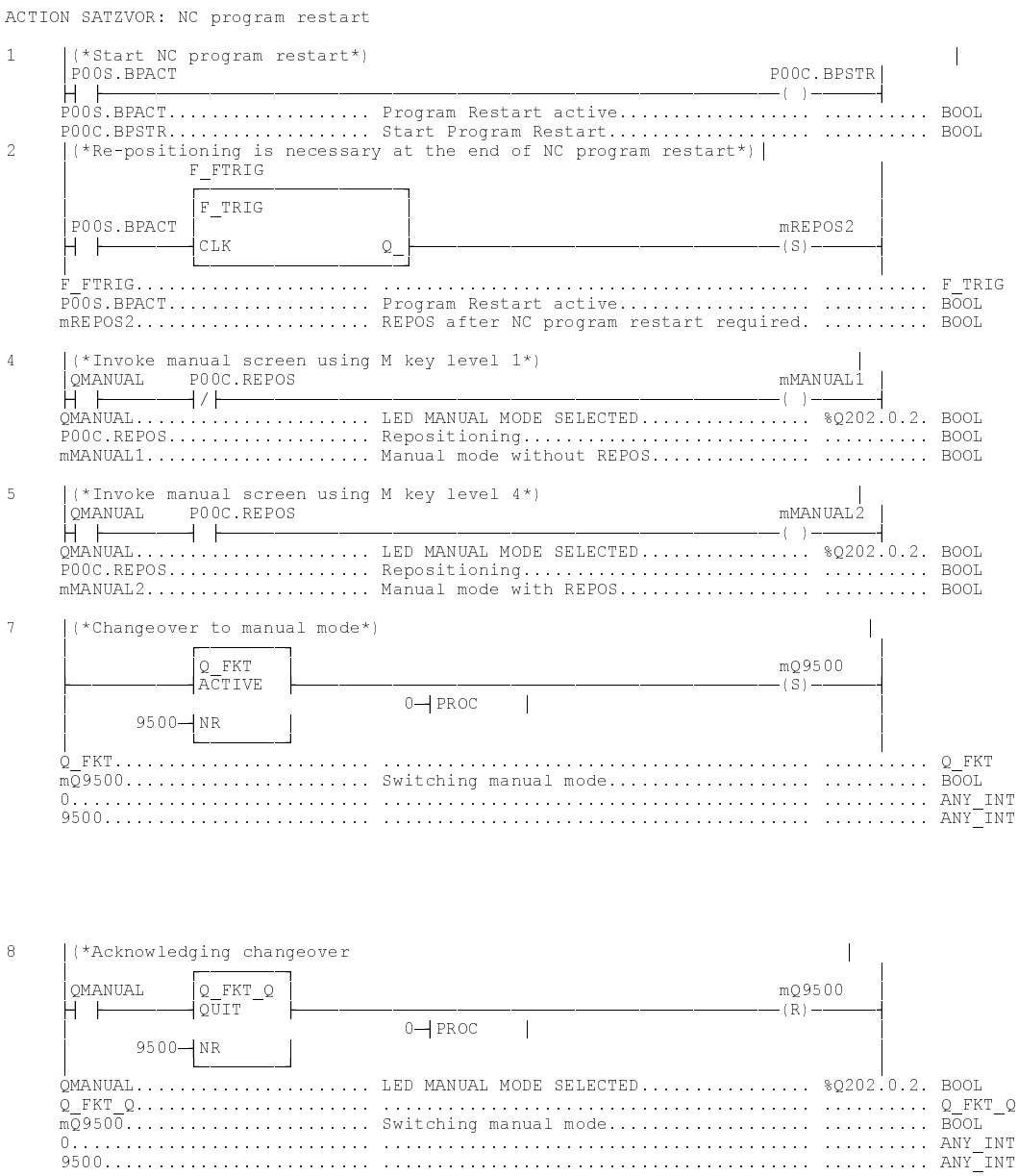

Figure 7-13: SPS program segment for NC program restart

Expansion in the REPOS-SPS program segment

An existing REPOS-SPS segment must be expanded in conjunction with NC program restart. In the following example, all major modifications are highlighted and marked with a ´>´.

REPOS

```
ACTION aREPOS: REPOSITIONING, RETURN TO CONTOUR

1     |(*Repositioning and return to contour permitted*) |
      |P00S.REPOS P00S.CREST mRuW_AUS                                        mRuW_OK  |
      ├┤ ├───────┤/├───────┤/├─────────────────────────────────────────────( )──────┤
      P00S.REPOS.................. Repositioning/Restart allowed.......... .......... BOOL
      P00S.CREST.................. Conditions restored.................... .......... BOOL
      mRuW_AUS.................... REPOS./RETURN TO CONTOUR OFF........... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour.. .......... BOOL

>  2  |(*Reset flag for REPOS after NC program restart                               |
>     |P00C.REPOS                                                            mREPOS2  |
>     ├┤ ├─────────────────────────────────────────────────────────────────(R)──────┤
>     P00C.REPOS.................. Repositioning.......................... .......... BOOL
>     mREPOS2..................... REPOS after NC program restart required. .......... BOOL

4     |(*REPOSITIONING*) |
      |mREPOS     mRuW_OK    mRESTA                                       P00C.REPOS|
      ├┤ ├───────┤ ├───────┤/├─────────────────────────────────────────────( )──────┤
>     |mREPOS2   |                                                                   |
>     ├┤ ├───────┤                                                                   |
      |P00C.REPOS|                                                                   |
      ├┤ ├───────┘                                                                   |
      mREPOS...................... REPOSITIONING.......................... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour.. .......... BOOL
      mRESTA...................... RETURN TO CONTOUR...................... .......... BOOL
      P00C.REPOS.................. Repositioning.......................... .......... BOOL
      mREPOS2..................... REPOS after NC program restart required. .......... BOOL
      P00C.REPOS.................. Repositioning.......................... .......... BOOL

5     |(*RETURN TO CONTOUR*) |
      |mRESTA     mRuW_OK    mREPOS                                         P00C.RESTA
      ├┤ ├───────┤ ├───────┤/├─────────────────────────────────────────────( )──────┤
      |P00C.RESTA|                                                                   |
      ├┤ ├───────┘                                                                   |
      mRESTA...................... RETURN TO CONTOUR...................... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour.. .......... BOOL
      mREPOS...................... REPOSITIONING.......................... .......... BOOL
      P00C.RESTA.................. Restart................................ .......... BOOL
      P00C.RESTA.................. Restart................................ .......... BOOL

6     |(*Triggering the screen*)                                                    |
      |P00C.REPOS                                                           mRuW_BLD |
      ├┤ ├─────────────────────────────────────────────────────────────────( )──────┤
      |P00C.RESTA|                                                                   |
      ├┤ ├───────┘                                                                   |
      P00C.REPOS.................. Repositioning.......................... .......... BOOL
      mRuW_BLD.................... BILDSCH.FUER R.+W...................... .......... BOOL
      P00C.RESTA.................. Restart................................ .......... BOOL

7     |(*ADJUSTMENT S1 M3*)                                                          |
      |mA04CM3    mRuW_OK    mSP1_ABGL                                      MREPOSM3 |
      ├┤ ├───────┤ ├───────┤ ├─────────────────────────────────────────────( )──────┤
      mA04CM3..................... ADJUSTMENT S1 M3......................... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour. .......... BOOL
      mSP1_ABGL................... Adjustment spindle 1.................. .......... BOOL
      MREPOSM3.................... M start spindle RePos M3.............. .......... BOOL


8     |(*ADJUSTMENT S1 M4*)                                                          |
      |mA04CM4    mRuW_OK    mSP1_ABGL                                      MREPOSM4 |
      ├┤ ├───────┤ ├───────┤ ├─────────────────────────────────────────────( )──────┤
      mA04CM4..................... ADJUSTMENT S1 M4...................... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour. .......... BOOL
      mSP1_ABGL................... Adjustment spindle 1.................. .......... BOOL
      MREPOSM4.................... M start spindle RePos M4.............. .......... BOOL

9     |(*ADJUSTMENT S1 M5*)                                                          |
      |mA04CM5    mRuW_OK    mSP1_ABGL                                      MREPOSM5 |
      ├┤ ├───────┤ ├───────┤ ├─────────────────────────────────────────────( )──────┤
      mA04CM5..................... ADJUSTMENT S1 M5...................... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour. .......... BOOL
      mSP1_ABGL................... Adjustment spindle 1.................. .......... BOOL
      MREPOSM5.................... M start spindle RePos M5.............. .......... BOOL

10    |(*ADJUSTMENT S1 M19*)                                                         |
      |mA04CM19   mRuW_OK    mSP1_ABGL                                      MREPOSM19|
      ├┤ ├───────┤ ├───────┤ ├─────────────────────────────────────────────( )──────┤

      mA04CM19.................... ADJUSTMENT S1 M19..................... .......... BOOL
      mRuW_OK..................... Enable repositioning/return to contour. .......... BOOL
      mSP1_ABGL................... Adjustment spindle 1.................. .......... BOOL
      MREPOSM19................... M start spindle RePos M19............. .......... BOOL
```

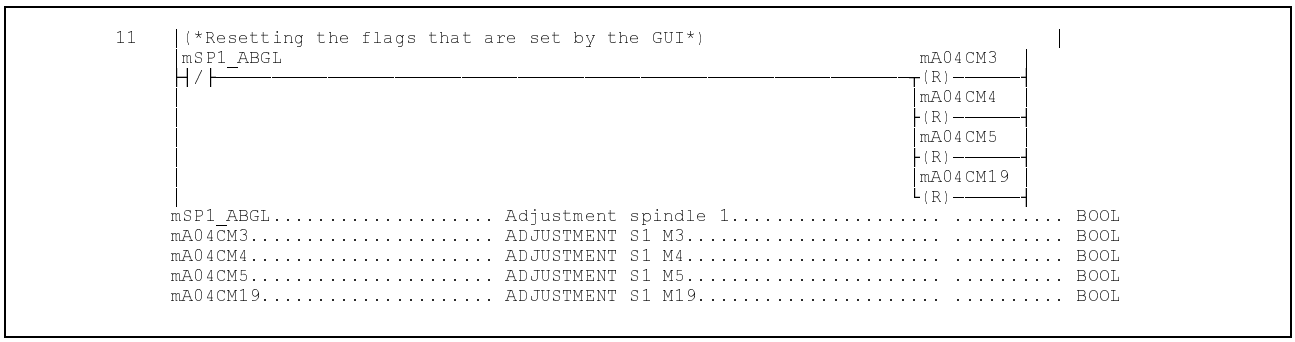

Figure 7-14: Expansion in the REPOS-SPS program segment

ADJUST and REPOS subroutines

ADJUST and REPOS subroutines

```
.ADJUST
 @127=G(3) @128=G(4) @129=G(6);Retaining tool path corr., zero
offset+dimens.
@130=M(2) @131=M(5)           ;Retaining spindle commands and coolant
@126=T                        ;Retaining selected (prepositioned) tool
T= MTD(60,,,5) BSR .M6        ;Changing in required tool
T=@126 MTP                    ;Prepositioning selected tool
G=@128 G=@129                 ;Setting stored zero offset and dimensions
G=@127                        ;Setting stored tool path correction
;
Q9500                         ;Changing over to manual mode
;
.REPOS
 G77 S0 MTP                   ;Adjusting spindle and magazine
                              (prepositioning selected tool)
MRY                           ;Waiting until magazine movement stops
;
; Setting auxiliary functions
M=@130 M=@131
;
;Adjustment of linear axes according to selected plane
@100=G(2)-18 BEQ .REPOS1   ;Plane
@100=G(2)-19 BEQ .REPOS2
G77 X0 Y0 F2000
G77 Z0 F2000 RTS
.REPOS1
 G77 Z0 X0 F2000
G77 Y0 F2000 RTS
.REPOS2
 G77 Y0 Z0 F2000
 G77 X0 F2000 RTS
```

Figure 7-15: ADJUST and REPOS subroutines

7.2 Turning Center

Task

NC program restart including multi-block retrace (repositioning) shall be commissioned for a turning center (see figure below).

Structure of the turning center

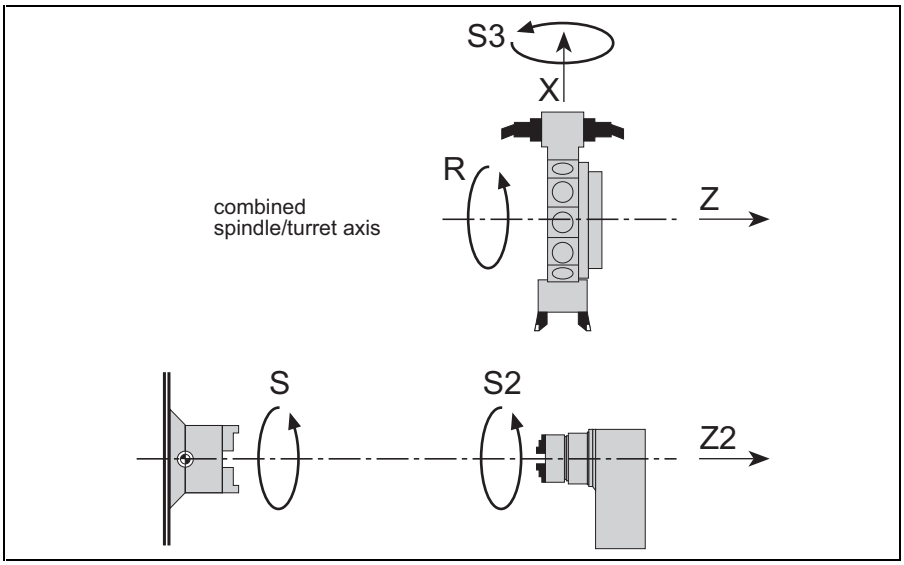

Figure 7-16: Structure of the turning center

The turning center mainly consists of a main spindle S and a counter spindle (S2) that is moved using a secondary axis Z2, and an X axis and a Z axis that move the turret. During the machining process, the workpieces are transferred from the first spindle (S) to the counter spindle (S2) where they are finished. Furthermore, there is a combined spindle/turret axis (R/S3) for the turret with 12 locations. Driven tools (twin tools) can be programmed using the address S3.

Parameter value assignments

Bxx.058	M function output during NC program restart	Yes
Bxx.059	Q function output during mid-program startup	No
Bxx.060	S function output during mid-program startup	No
Bxx.061	T/E function output during mid-program startup	No

Parts transfer is implemented on the machine data page, in the SPS program, in the ADJUST and REPOS subroutines, and in the subroutine for the parts transfer (see figures below) .

Machine data page for parts transfer

The ´direction S1<>S2´ and ´Position Z2 axis´ elements are accessed via the machine data page.

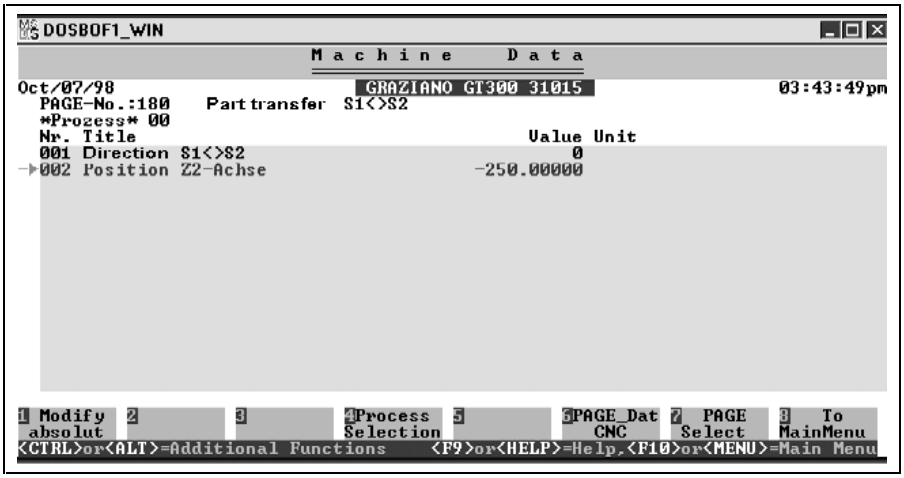

Figure 7-17: Machine data page for parts transfer

Function block used for initializing the parts transfer

The MTD_WR function block is used for writing the machine data for the part transfer in the SPS program.

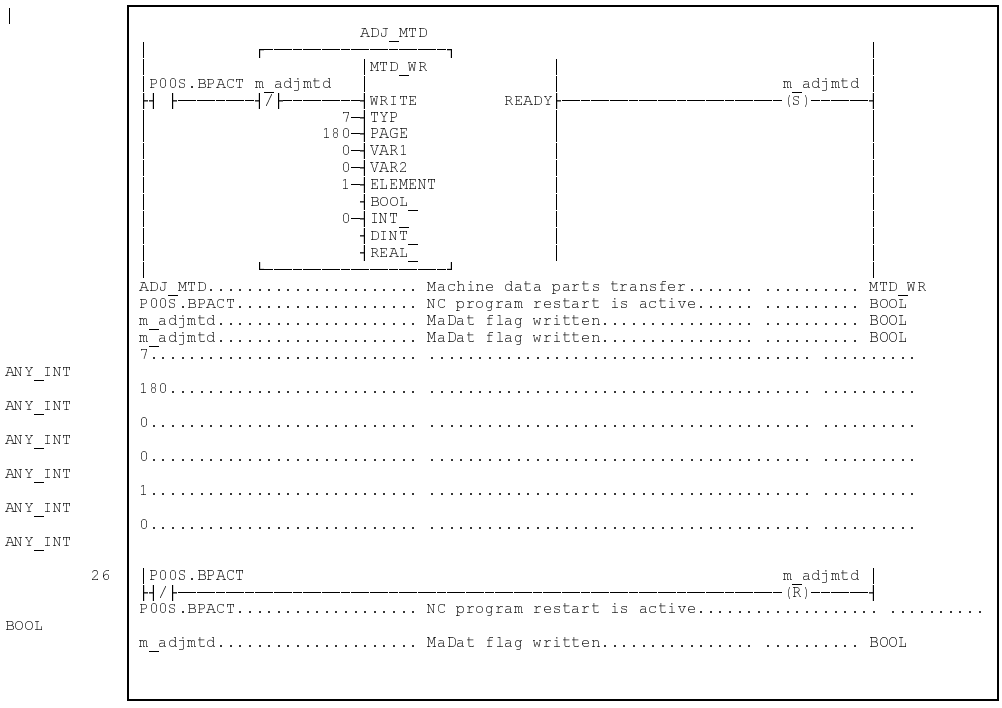

Figure 7-18: SPS function block for parts transfer

ADJUST and REPOS subroutines

The states required for program entry are established in the ADJUST and REPOS subroutines. The workpiece transfer between spindle 1 and spindle 2 is processed in the 'parts transfer' subroutine.

```
%MAC:0:03:0:030
!01.00
#
$
$
$
$
*ADJUST - REPOS - RESTA
%MAC
;--------------------------- .ADJUST -------------------------------
N0000 .ADJUST
; Default values for ADJUST and REPOS
N0001 @105=0                            ; Tracking flag
N0002 @106=0                            ; Spindle synchronization flag
;


;------------------- Changing tools --------------------------------
N0003 @100=M(4)
N0004 T=MTD(60,,,5)                     ; Reading tool number
N0005 MTP                               ; Changing tools
N0006 MRY
N0007 M=@100
;
;------------------ Parts transfer S1<>S2 --------------------------
N0008 @100=MTD(180,,,1) BEQ .KWSTWE  ; Parts transfer necessary
N0009 @100=@141 @101=@142            ; Buffering the variables and
N0010 @102=S @103=S2                 ; spindle speeds fir parts transfer
N0011 @141=MTD(180,,,1)              ; Reading direction and position from
N0012 @142=MTD(180,,,2)              ; machine data user page
N0013 BSR .*WSTCH                    ; Parts transfer subroutine call
N0014 @141=@100 @142=@101            ; Backup of variablea and
N0015 S=@102 S2=@103                 ; spindle speeds
N0016 .KWSTWE
;
;------------------ Output of M auxiliary functions ----------------
N0017 @100=1                          ; Count variable
N0018 .LOOP1
N0019 @110=MTD(61,,1,@100) BEQ .WEIT1 ; Read from machine data
; M group 16
;        Skipping M functions that are nor executed
N0020 @101=@110-68 BEQ .MF_NAU
N0021 @101=@110-69 BEQ .MF_NAU
N0022 @101=@110-268 BEQ .MF_NAU
N0023 @101=@110-269 BEQ .MF_NAU
;
;        Considering spindle synchronization
N0024 @101=@110-70 BNE .SSKAN
N0025 @106=@110                       ; Spindle sync flagON
N0026 BRA .MF_NAU                     ; Don't execute M function
```

```
N0027 .SSKAN
N0028 @101=@110-71 BNE .SSKAUS
N0029 @106=@110                          ; Spindle sync  flagOFF
N0030 BRA .MF_NAU                        ; Don't execute M function
N0031 .SSKAUS
;
;        Considering tracking
N0032 @101=@110-270 BNE .MUGAN
N0033 @105=@110                          ; Tracking flag ON
N0034 BRA .MF_NAU                        ; Don't execute M function
N0035 .MUGAN
N0036 @101=@110-271 BNE .MUGAUS
N0037 @105=@110                          ; Tracking flag OFF
N0038 BRA .MF_NAU                        ; Don't execute M function
N0039 .MUGAUS
;
N0040 M=@110                            ; Executing remaining M functions
                                        ; of group 16
N0041 .MF_NAU
N0042 @100=@100+1
N0043 @101=@100-41 BMI .LOOP1
N0044 .WEIT1
;
;------------------ Output of Q auxiliary functions ----------------------
N0045 @100=1                             ; Count variable
N0046 .LOOP2
N0047 @110=MTD(61,,2,@100) BEQ .WEIT2  ; Read from machine data
; Q group 16
N0048 Q=@110                             ; Executing Q functions of
                                         ; group 16
N0049 @100=@100+1
N0050 @101=@100-41 BMI .LOOP2
N0051 .WEIT2
;
;----------- Setting G code group 17 (transformation) --------------------
;
N0052 @107=G(1) @108=G(3)              ; Buffering the active
interpolation,
N0053 @109=G(4) @110=G(13)             ; tool path compensation
                                       ; Zero point shifts
                                       ; Tool length compensation
N0054 @101=MTD(60,,,4)-32 BEQ .G32TRA ; G32 interrogation active
N0055 G=@101+32                        ; Setting G30 or G31
N0056 BRA .WEIT4
;
;        Setting prerequisites for  G32
N0057 .G32TRA
N0058 @100=SPC-1 BEQ .G32SP1           ; Interrogating the reference
spindle
                                       ; Reference spindle 2
N0059 G18                              ; Resetting G20
N0060 @100=G(6) @101=G(1)              ; Buffering the G groups
                                       ; Interpolation, dimension
N0061 G91 G00 C2 0                     ; Switching to rotary axis mode
N0062 G=@100 G=@101                    ; Backup of G groups
```

```
N0063 G20 Z0 C2 0 X0                    ; Setting G20 for spindle 2
N0064 BRA .G32UMS
N0065 .G32SP1                           ; Reference spindle 1
N0066 G18                               ; Resetting G20
N0067 @100=G(6) @101=G(1)               ; Buffering the G groups
                                        ; Interpolation, dimension
N0068 G91 G00 C0                        ; Switching to rotary axis mode
N0069 G=@100 G=@101                     ; Backup of G groups
N0070 G20 Z0 C0 X0                      ; Setting G20 for spindle 1
N0071 .G32UMS
N0072 G32                               ; Activating G32
;
N0073 .WEIT4
N0074 G=@107 G=@108                     ; Backup of G groups
N0075 G=@110                            ; G(1), G(3), G(4), G(13)
N0076 G=@109
;
;        Deactivating tracking before REPOS
N0077 @101=@105-270 BNE .KEIMG3         ; Tracking interrogation active
N0078 M271                              ; Tracking OFF
N0079 .KEIMG3
;
;-------------- Switching to manual mode-----------------------------
N0080 Q9500                             ; Switch to manual mode via SPS
;
; ===================================================================
;---------------------------- .REPOS --------------------------------
N0081 .REPOS
;
;-------------- Continue in automatic mode --------------------------
;
;-------------------- Repositioning spindle 3 ----------------------
;
N0082 @100=M(4)-305 BEQ .SP3END
N0083 G77 S3 0
N0084 .SP3END
;
N0085 @101=@106-71 BNE .NSYAUS          ; Switch off sync interrogation
N0086 M71
N0087 .NSYAUS
;
;------------ Spindle positioning depending G(17) ------------------
;
N0088 @100=MTD(60,,,4)-30 BEQ .WEIT11 ; G30 interrogation active
;
N0089 @101=SPC-1 BEQ .BEZSP2            ; Interrogation transformation
spindle 1
;
N0090 @101=MTD(60,,,1) BEQ .SP1POS      ; Approaching spindle 1
N0091 G77 C0 F5000                      ; Spindle 1 in rotary axis mode
N0092 BRA .TRANS2
N0093 .SP1POS
N0094 G77 S0                            ; Spindle 1 in spindle mode
;
```

```
; ------------- Transformation spindle 2 ----------------------------
N0095 .TRANS2
N0096 @101=MTD(60,,,4)-31 BEQ .S2_G32 ; G32 interogation active
;
N0097 G77 X0 Y2 0 F5000               ; Adjustment of spindle 2 for G31
N0098 G77 Z0 F5000                    ; Adjustment of Z axis
N0099 G77 Z2 0 F5000                  ; Adjustment of Z2 axis
N0100 BRA .TRAAKT
;
N0101 .S2_G32                         ; Adjustment of spindle 2 for G32
N0102 G77 Y2 0 F5000                  ; Adjustment of Y2 axis
N0103 G77 Z0 F5000                    ; Adjustment of Z axis
N0104 G77 Z2 0 F5000                  ; Adjustment of Z2 axis
N0105 G77 X0 F5000                    ; Adjustment of X axis
;
N0106 BRA .TRAAKT
;
N0107 .BEZSP2
N0108 @101=MTD(60,,,2) BEQ .SP2POS    ; Approaching spindle 2 position
N0109 G77 C2 0 F5000                  ; Spindle 2 in rotary axis mode
N0110 BRA .TRANS1
N0111 .SP2POS
N0112 G77 S2 0                        ; Spindle 2 in spindle mode
;
; ------------- Transformation spindle 1 ----------------------------
N0113 .TRANS1
N0114 @101=MTD(60,,,4)-32 BEQ .S1_G32 ; G32 interrogation active
;
N0115 G77 X0 Y1 0 F5000               ; Adjustment of spindle 1 for G31
N0116 G77 Z0 F5000                    ; Adjustment of Z axis
N0117 G77 Z2 0 F5000                  ; Adjustment of Z2 axis
N0118 BRA .TRAAKT
;
N0119 .S1_G32                         ; Adjustment of spindle 1 for G32
N0120 G77 Y1 0 F5000                  ; Adjustment of Y1 axis
N0121 G77 Z0 F5000                    ; Adjustment of Z axis
N0122 G77 Z2 0 F5000                  ; Adjustment of Z2 axis
N0123 G77 X0 F5000                    ; Adjustment of X axis
;
N0124 BRA .TRAAKT
N0125 .WEIT11
;
;------------------- Adjusting spindle 1 and spindle 2 -------------------
--
;
N0126 @100=MTD(60,,,1)+MTD(60,,,2) BEQ .SPMOD ; Both spindles in S mode
;
N0127 @101=MTD(60,,,1) BEQ .S1SMOD    ; Approaching spindle 1 position
N0128 G77 C0 F5000                    ; Spindle 1 in rotary axis mode
N0129 BRA .S2ABG
N0130 .S1SMOD
N0131 G77 S0                          ; Spindle 1 in spindle mode
;
N0132 .S2ABG
```

```
N0133 @101=MTD(60,,,2) BEQ .S2SMOD    ; Approaching spindle 2 position
N0134 G77 C2 0 F5000                  ; Spindle 2 in rotary axis mode
N0135 BRA .RUMOD
N0136 .S2SMOD
N0137 G77 S2 0                        ; Spindle 2 in spindle mode
N0138 BRA .RUMOD
;
N0139 .SPMOD
;
N0140 @101=@106-70 BNE .NSYNCH        ; Activating sync interrogation
N0141 M=M(2)                          ; Spindle 1 in S mode
N0142 M=M(3)                          ; Spindle 2 in S mode
N0143 G77 S0 S2 0 M70                 ; Establishing synchronization
N0144 BRA .RUMOD
N0145 .NSYNCH
N0146 G77 S0 S2 0                     ; Adjustment spindle 1 and spindle 2
;
N0147 .RUMOD
;
;----------------- Repositioning axes -------------------------------
;
N0148 G77 X0 F5000                    ; Adjustment of X axis
N0149 G77 Z0 F5000                    ; Adjustment of Z axis
N0150 G77 Z2 0 F5000                  ; Adjustment of Z2 axis
;
N0151 .TRAAKT
;
N0152 @101=@105-270 BNE .KEIMG2       ; Tracking interrogation active
N0153 M270                            ; Tracking ON
N0154 .KEIMG2
;
;
N0155 RTS
N0156 RTS
```

Figure 7-19: ADJUST and REPOS subroutines

NC subroutine for parts transfer

```
%MAC:0:03:0:019
!01.00
#
$
$
$
*Parts transfer S1<>S2
%MAC
%CHBEGIN%
%V% @141 INT[1,2] parts transfer S2>S1 (1) S1>S2 (2)
%V% @142 REAL[-450,-0.1] transfer position Z2 axis
%N% BSR .*WSTCH  ;cycle call
%CHEND%
;*******************************
;***   Cycle : Cycle name   ***
;***   Label : .xxxxx        ***
;*******************************
```

```
;
N0000 .*WSTCH
N0001 MTD(180,,,1)=@141  ; Direction
N0002 MTD(180,,,2)=@142  ; Position
N0003 @101=G(4)
;
N0004 @100=@141-2 BEQ .*WSTS2
; Part transferred in 1st spindle
N0005 M=69
N0006 G4 F.2
N0007 M10
N0008 S20 M3
N0009 G4 F2
N0010 M5
N0011 M11
N0012 G4 F.2
N0013 G53 G0 Z2=@142
N0014 M=68
N0015 G4 F1
N0016 M269
N0017 G4 F.5
N0018 G0 Z2 1
N0019 BRA .*WSTEN
;
N0020 .*WSTS2
; Part transferred in spindle 2
N0021 M269
N0022 G4 F.2
N0023 M10
N0024 S2 20 M203
N0025 G4 F2
N0026 M205
N0027 M11
N0028 G4 F.2
N0029 G53 G0 Z2=@142
N0030 M268
N0031 G4 F1
N0032 M=69
N0033 G4 F.5
N0034 G0 Z2 1
;
N0035 .*WSTEN
N0036 G=@101
;
N0037 RTS
N0038 RTS
```

Figure 7-20: NC subroutine for parts transfer

Kundenbetreuungsstellen - Sales & Service Facilities

Deutschland - Germany

Germany Center ☒V/S ☒ Service	Vertriebsgebiet Ost Germany East ☒V/S ☒ Service	Vertriebsgebiet West Germany West ☒V/S ☒ Service	Vertriebsgebiet Nord Germany North ☒V/S ☐ Service
INDRAMAT GmbH Bgm.-Dr.-Nebel-Str. 2 D - 97816 Lohr am Main Phone: +49 (0)9352/40-0 Fax: +49 (0)9352/40-4885	INDRAMAT GmbH Beckerstraße 31 D - 09120 Chemnitz Phone: +49 (0)371/35 55-0 Fax: +49 (0)371/35 55-333	INDRAMAT GmbH Harkortstraße 25 D - 40849 Ratingen Phone: +49 (0)2102/43 18-0 Fax: +49 (0)2102/41 315	INDRAMAT GmbH Kieler Straße 212 D - 22525 Hamburg Phone: +49 (0)40/85 31 57-0 Fax: +49 (0)40/85 31 57-15
Vertriebsgebiet Süd Germany South ☒V/S ☐ Service	Gebiet Südwest Germany South-West ☒V/S ☒ Service		INDRAMAT Service-Hotline
INDRAMAT GmbH Ridlerstraße 75 D-80339 München Phone: +49 (0)89/540138-30 Fax: +49 (0)89/540138-10	INDRAMAT GmbH Böblinger Straße 25 D-71229 Leonberg Phone: +49 (0)7152/9 72-6 Fax: +49 (0)7152/9 72-727		INDRAMAT GmbH Phone: +49 (0)172/660 04 06 or Phone: +49 (0)171/333 88 26

Kundenbetreuungsstellen in Deutschland - Service agencies in Germany

Europa - Europe

Austria ☒V/S ☐ Service Mannesmann Rexroth Ges.m.b.H. Geschäftsbereich INDRAMAT Hägelingasse 3 A - 1140 Wien Phone: +43 (0)1/9852540-400 Fax: +43 (0)1/9852540-93	Austria ☐V/S ☒ Service Mannesmann Rexroth G.m.b.H. Geschäftsbereich INDRAMAT Industriepark 18 A - 4061 Pasching Phone: +43 (0)7221/605-0 Fax: +43 (0)7221/605-21	Belgium ☒V/S ☒ Service Mannesmann Rexroth N.V.-S.A. Geschäftsbereich INDRAMAT Industrielaan 8 B-1740 Ternat Phone: +32 (0)2/5823180 Fax: +32 (0)2/5824310	Denmark ☒V/S ☒ Service BEC AS Zinkvej 6 DK-8900 Randers Phone: +45 (0)87/11 90 60 Fax: +45 (0)87/11 90 61
United Kingdom ☒V/S ☒ Service Mannesmann Rexroth Ltd. INDRAMAT Division 4 Esland Place, Love Lane GB - Cirencester, Glos GL7 1YG Phone: +44 (0)1285/658671 Fax: +44 (0)1285/654991Ł ł	Finland ☒V/S ☐ Service Rexroth Mecman OY Ansatie 6 SF-017 40 Vantaa Phone: +358 (0)9/84 91 11 Fax: +358 (0)9/84 91 13 60	France ☒V/S ☒ Service Mannesmann Rexroth S.A. Division INDRAMAT Parc des Barbanniers 4, Place du Village F-92632 Gennevilliers Cedex Phone: +33 (0)141 47 54 30 Fax: +33 (0)147 94 69 41 Hotline: +33 (0)6 08 33 43 28	France ☒V/S ☐ Service Mannesmann Rexroth S.A. Division INDRAMAT 270, Avenue de Lardenne F - 31100 Toulouse Phone: +33 (0)5 61 49 95 19 Fax: +33 (0)5 61 31 00 41
France ☒V/S ☐ Service Mannesmann Rexroth S.A. Division INDRAMAT 91, Bd. Irène Joliot-Curie F - 69634 Vénissieux Cedex Phone: +33 (0)4 78 78 53 65 Fax: +33 (0)4 78 78 52 53	Italy ☒V/S ☒ Service Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via G. Di Vittoria, 1 I - 20063 Cernusco S/N.MI Phone: +39 (0)2/92 36 52 70 Fax: +39 (0)2/92 36 55 12Ł ł	Italy ☒V/S ☒ Service Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via Borgomanero, 11 I - 10145 Torino Phone: +39 (0)11/7 71 22 30 Fax: +39 (0)11/7 71 01 90	Italy ☒V/S ☐ Service Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via del Progresso, 16 (Zona Ind.) I - 35020 Padova Phone: +39 (0)49/8 70 13 70 Fax: +39 (0)49/8 70 13 77
Italy ☐V/S ☒ Service Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via de Nicola, 12 Ł ł I - 80053 Castellamare di Stabbia NAŁ ł Phone: +39 (0)81/8 72 30 37 Fax: +39 (0)81/8 72 30 18	Italy ☒V/S ☐ Service Mannesmann Rexroth S.p.A. Divisione INDRAMAT Viale Oriani, 38/A I - 40137 Bologna Phone: +39 (0)51/34 14 14 Fax: +39 (0)51/34 14 22	Netherlands ☒V/S ☐ Service Hydraudyne Hydrauliek B.V. Kruisbroeksestraat 1 P.O. Box 32 NL - 5281 RV Boxtel Phone: +31 (0)411/65 19 51 Fax: +31 (0)411/65 14 83 Ł ł	Netherlands ☐V/S ☒ Service Hydrocare B.V. Kruisbroeksestraat 1 P.O. Box 32 NL - 5281 RV Boxtel Phone: +31 (0)411/65 19 51 Fax: +31 (0)411/67 78 14 Ł ł
Spain ☒V/S ☒ Service Mannesmann Rexroth S.A. Divisiòn INDRAMAT Centro Industrial Santiga Obradors s/n Ł ł E-08130 Santa Perpetua de Mogoda Ł łBarcelona Phone: +34 937 47 94 00 Fax: +34 937 47 94 01	Spain ☒V/S ☒ Service Goimendi S.A. División Indramat Jolastokieta (Herrera) Apartado 11 37 E - 20017 San Sebastian Phone: +34 9 43/40 01 63 Fax: +34 9 43/39 17 99	Sweden ☒V/S ☒ Service Rexroth Mecman Svenska AB INDRAMAT Division Varuvägen 7 S - 125 81 Stockholm Phone: +46 (0)8/727 92 00 Fax: +46 (0)8/64 73 277	Switzerland - East ☒V/S ☒ Service Mannesmann Rexroth AG Geschäftsbereich INDRAMAT Gewerbestraße 3 CH-8500 Frauenfeld Phone: +41 (0)52/720 21 00 Fax: +41 (0)52/720 21 11
Switzerland - West ☒V/S ☐ Service Mannesmann Rexroth SA Département INDRAMAT Chemin de l`Ecole 6 CH-1036 Sullens Phone: +41 (0)21/731 43 77 Fax: +41 (0)21/731 46 78	Russia ☐V/S ☒ Service Tschudnenko E.B. Arsenia 22 RUS - 153000 Ivanovo Rußland Phone: +7 093/223 96 33 Ł łoder/orŁ ł +7 093/223 95 48 Fax: +7 093/223 46 01	Slowenia ☒V/S ☒ Service DOMEL Elektromotorji in gospodinjski aparati d. d. Otoki 21 SLO - 64 228 Zelezniki Phone: +386 64/61 73 32 Fax: +386 64/64 71 50	Turkey ☒V/S ☒ Service Ł ł Mannesmann Rexroth Hidropar A..S.Ł ł Fevzi Cakmak Cad No. 3 TR - 34630 Sefaköy Istanbul Phone: +90 212/541 60 70 Fax: +90 212/599 34 07

Europäische Kundenbetreuungsstellen (ohne Deutschland)
European Service agencies (without Germany)

Außerhalb Europa - Outside Europe

Argentina ☒V/S ☐ Service Mannesmann Rexroth S.A.I.C. Division INDRAMAT Acassusso 48 41/7 RA - 1605 Munro (Buenos Aires) Phone: +54 (0)1/756 01 40 +54 (0)1/756 01 36	Argentina ☐V/S ☒ Service Ł ŀNAKASEŁ ŀ Asesoramiento Tecnico Calle 49, No. 5764-66 RA - 1653 Villa Balester Provincia de Buenos Aires Phone: +54 (0) 1/768 24 13 Fax: +54 (0) 1/768 36 43 ŀ	Australia ☒V/S ☒ Service AIMS - Australian Industrial Machinery Services Pty. Ltd. Unit 3/45 Horne ST Campbellfield 3061 AUS - Melbourne, VIC Phone: +61 (0)3/93 59 02 28 Fax: +61 (0)3/93 59 02 86 ŀ	Brazil ☒V/S ☒ Service Mannesmann Rexroth Automação Ltda. Divisão INDRAMAT Rua Georg Rexroth, 609 Vila Padre Anchieta BR - 09951-270 Diadema-SP [Ł ŀCaixa Postal 377] [BR-09901-970 Diadema-SP]Ł ŀ Phone: +55 (0)11/745 90 60 +55 (0)11/745 90 70 Fax: +55 (0)11/745 90 50
Canada ☒V/S ☒ Service Basic Technologies Corporation Burlington Division 3426 Mainway Drive Burlington, Ontario Canada L7M 1A8 Phone: +1 905/335 55 11 Fax: +1 905/335-41 84	China ☒V/S ☐ Service Mannesmann Rexroth (China) Ldt. Shanghai Office - Room 206 Shanghai Internat. Trade Centre 2200 Yanan Xi Lu PRC - Shanghai 200335 Phone: +86 21/62 75 53 33 Fax: +86 21/62 75 56 66 Ł ŀ	China ☐V/S ☒ Service Mannesmann Rexroth (China) Ldt. Shanghai Parts & Service Center 199 Wu Cao Road, Hua Cao Minhang District PRC - Shanghai 201 103 Phone: +86 21/62 20 00 58 Fax: +86 21/62 20 00 68 Ł ŀ	China ☒V/S ☐ Service Mannesmann Rexroth (China) Ldt. 15/F China World Trade Center 1, Jianguomenwai Avenue PRC - Beijing 100004 Phone: +86 10/65 05 03 80 Fax: +86 10/65 05 03 79 Ł ŀ
China ☒V/S ☐ Service Mannesmann Rexroth (China) Ldt. A-5F., 123 Lian Shan Street Sha He Kou District PRC - Dalian 116 023 Phone: +86 411/46 78 930 Fax: +86 411/46 78 932	Hongkong ☒V/S ☒ Service Rexroth (China) Ldt. 19 Cheung Shun Street 1ŁłstŁł Floor, Cheung Sha Wan, Kowloon, Hongkong Phone: +852 27/41 13 51/-54 Ł łoder/orŁ ŀ +852 27/41 14 30 Fax: +852 27/86 07 33 Ł ŀ	India ☒V/S ☒ Service Mannesmann Rexroth (India) Ltd. INDRAMAT Division Plot. 96, Phase III Peenya Industrial Area IND - Bangalore - 560058 Phone: +91 (0)80/8 39 21 01 Fax: +91 (0)80/8 39 43 45 Ł	India ☒V/S ☒ Service Mannesmann Rexroth (India) Ltd. INDRAMAT Division Plot. A-58, TTC Industrial Area Thane Turbhe Midc Road Mahape Village IND - Navi Mumbai - 400 701 Phone: +91 (0)22/7 61 46 22 Fax: +91 (0)22/7 68 15 31
Indonesia ☒V/S ☐ Service PT. Rexroth Wijayakusuma Jl. Raya Bekasi Km 21 Pulogadung RI - Jakarta Timur 13920 Phone: +62 21/4 61 04 87 +62 21/4 61 04 88 Fax: +62 21/4 60 01 52	Japan ☒V/S ☒ Service Rexroth Automation Co., Ltd. INDRAMAT Division 1F, I.R. Building Nakamachidai 4-26-44 Tsuzuki-ku, Yokohama-shi J - Kanagawa-ken 224-004 Phone: +81 459/42-72 10 Fax: +81 459/42-03 41	Korea ☒V/S ☒Service Ł ŀ Mannesmann Rexroth-Seki Co Ltd.Ł ŀ 1500-12 Da-Dae-Dong ROK - Saha-Ku, Pusan, 604-050 Phone: +82 (0)51/2 60 06 18 Fax: +82 (0)51/2 60 06 19 ŀ	Korea ☒V/S ☒ Service Seo Chang Corporation Ltd. Room 903, Jeail Building 44-35 Yeouido-Dong Yeoungdeungpo-Ku C.P.O.Box 97 56 ROK - Seoul Phone: +82 (0)2/7 80 82 08 +82 (0)2/7 80 82 09 Fax: +82 (0)2/7 84 54 08
Mexico ☒V/S ☐Service Rexroth Mexico S.A. de C.V. Calle Neptuno 72 Unidad Ind. Vallejo MEX - 07700 Mexico, D.F. Phone: +52 5 754 17 11 +52 5 754 36 84 +52 5 754 12 60 Fax: +52 5 754 50 73 +52 5 752 59 43 Ł ŀ		South Africa ☒V/S ☒ Service HYTEC Automation (Pty) Ltd. 28 Banfield Road,Industria North RSA - Maraisburg 1700 Phone: +27 (0)11/673 20 80 Fax: +27 (0)11/673 72 69 Ł ŀ	Taiwan ☒V/S ☐ Service Rexroth Uchida Co., Ltd. No.1, Tsu Chiang Street Tu Cheng Ind. Estate Taipei Hsien, Taiwan, R.O.C. Phone: +886 2/2 68 13 47 Fax: +886 2/2 68 53 88

Kundenbetreuungsstellen außerhalb Europa - Service agencies outside Europe

Außerhalb Europa / USA - Outside Europe / USA

USA ☒ V/S ☒ Service

Mannesmann Rexroth Corporation
INDRAMAT Division
5150 Prairie Stone Parkway
Ł ł USA Ł ł Hoffman Estates Ł ł , IL
60192-3707 Ł ł

Phone: +1 847/6 45 36 00
Fax: +1 847/6 45 62 01

USA ☒ V/S ☒ Service Ł

Mannesmann Rexroth Corporation
INDRAMAT Division
Central Region Technical Center
USA - Auburn Hills, MI 48326

Phone: +1 248/3 93 33 30
Fax: +1 248/3 93 29 06

USA ☒ V/S ☒ Service

Mannesmann Rexroth Corporation
INDRAMAT Division
Southeastern Technical Center
3625 Swiftwater Park Drive
USA - Suwanee
Georgia 30174

Phone: +1 770/9 32 32 00
+1 770/9 32 19 03

USA ☒ V/S ☒ Service

Mannesmann Rexroth Corporation
INDRAMAT Division
Northeastern Technical Center
99 Rainbow Road
USA - East Granby,
Connecticut 06026

Phone: +1 860/8 44 83 77
+1 860/8 44 85 95

USA ☒ V/S ☒ Service

Mannesmann Rexroth Corporation
INDRAMAT Division
Charlotte Regional Sales Office
14001 South Lakes Drive
USA - Charlotte,
North Carolina 28273

Phone: +1 704/5 83 97 62
+1 704/5 83 14 86

Kundenbetreuungsstellen außerhalb Europa / USA
Service agencies outside Europe / USA

Notes


Indramat