Description of Functions 01/2002 Edition

sinumerik

SINUMERIK 802S
SINUMERIK 802C

SIEMENS

SIEMENS

SINUMERIK 802S
SINUMERIK 802C

Description of Functions

Applies to

Control system	*Software version*
SINUMERIK 802S	3
SINUMERIK 802C	3

01.2002 Edition

Axis/ Spindle Monitoring 1

Continuous–Path Control, Exact Stop 2

Velocity, Setpoint/ Actual–Value System, ... 3

Manual Traversing and Handwheel Traversing 4

Program Mode 5

Compensation 6

Transversal Axis 7

Reference–Point Approach 8

Spindle 9

Output of Auxiliary Functions to PLC 10

Feeds 11

Tool Compensation 12

EMERGENCY STOP 13

Diverse Interface Signals 14

List of Interface Signals 15

SINUMERIK® Documentation

Printing history

Brief details of this edition and previous editions are listed below.

IThe status of each edition is shown by the code in the ”Remarks” column.

Status code in the “Remarks” column:

A . . . . . New documentation.
B . . . . . Unrevised reprint with new Order No.
C . . . . . Revised edition with new status.
If actual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on the page.

Edition	Order–No.	Remark
02.99	6FC5597–2AA10–0BP1	**A**
04.00	6FC5597–3AA10–0BP1	**C**
01.02	6FC5597–3AA10–0BP2	**C**

This Manual is included on the documentation on CD–ROM **(DOCONCD)**

Edition	Order–No.	Remark
11.02	6FC5298–6CA00–0BG3	**C**

Trademarks
SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are registered trademarks of Siemens. Third parties using for their own purposes any other names in this document which refer to trademarks might infringe upon the rights of trademark owners.

This publication was produced with Interleaf V 7

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for demages. All rights, including rights created by patent grant or registration of utility model or design, are reserved.

© Siemens AG 2002. All rights reserved.

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

We have checked that the contents of this document correspond to the hardware and software described. Nonetheless, differences might exist and therefore we cannot guarantee that they are completely identical. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition. We welcome suggestions for improvement.

Subject to change without prior notice.

Bestell–Nr. 6FC5597–3AA10–0BP2
Printed in the Federal Republic of Germany

Siemens–Aktiengesellschaft

Preface

The present documentation describes the functionalities provided by the control system SINUMERIK 802S/C in detail.

Objective

The Functions descriptions provide the information required for configuration and start–up.

Target group

The Functional Descriptions provide information for:

- the planning engineer of the equipment
- the PLC programmer when developing the PLC user program with the listed signals
- the start–up engineer after planning and designing of the equipment
- the serviceman for checking and interpreting the status displays and alarms.

Safety notices

This Manual contains notices intended to ensure your personal safety and to avoid material damage. The notices are highlighted by a warning triangle and, depending on the degree of hazard, represented as shown below:

Danger

indicates that loss of life, severe personal injury or substantial material damage **will** result if the appropriate precautions are not taken.

Warning

indicates that loss of life, severe personal injury or substantial material damage **may** result if the appropriate precautions are not taken.

Caution

indicates that minor personal injury or material damage may result if the appropriate precautions are not taken.

Caution

without a warning triangle means that a material damage **can** occur if the appropriate precautions are not taken.

Attention

means that an undesirede event or status **can** occur if the appropriate note is not observed.

Note

is used to draw your special attention to an important information on the product, the handling of the product or the corresponding part of the documentation.

Qualified personnel

Start–up and operation of a device may only be carried out by **qualified personnel**. Qualified personnel as referred to in the safety notices provided in this Manual are persons who are authorized to start up, ground and tag devices, systems and circuits according to the relevant safety standards.

Usage as per intended purpose

Please observe the following:

Warning

The device may only be used for the cases of application, as intended by the Catalog, and only in conjunction with third–party devices and components recommended or approved by Siemens.

The proper and safe operation of the product requires transport, storage and installation according to the relevant instructions and qualified operation and maintenance at the prescribed intervals.

Contents

1 Axis/Spindle Monitoring 1-11

1.1 Motion Monitoring Functions 1-12
1.1.1 Contour Monitoring 1-12
1.1.2 Positioning Monitoring 1-13
1.1.3 Zero–Speed Control 1-14
1.1.4 Clamping Monitoring 1-14
1.1.5 Set–Speed Monitoring 1-15
1.1.6 Actual Velocity Monitoring 1-15

1.2 Encoder Monitoring Functions 1-17
1.2.1 Encoder Limit Frequency Monitoring 1-17
1.2.2 Zero Mark Monitoring 1-17
1.2.3 Hardware Errors 1-18

1.3 Limit Switch Monitoring 1-19

1.4 Axis Monitoring Supplementary Conditions 1-20

1.5 Stepper Motor Rotation Monitoring Using BERO 1-21

1.6 Data Description 1-22

1.7 Signal Description 1-28

2 Continuous–Path Control, Exact Stop 2-31

2.1 General 2-32
2.1.1 Velocities 2-32
2.1.2 Stopping for Synchronization 2-33

2.2 Exact Stop 2-34

2.3 Continuous–Path Control Mode 2-36

2.4 Data Description 2-37

2.5 Signal Description 2-38

3 Velocity, Setpoint/Actual–Value System, Closed–Loop Control 3-39

3.1 Setpoint/Actual–Value System 3-42
3.1.1 General 3-42
3.1.2 Speed Setpoint Output and Actual–Value Processing 3-43

3.2 Closed–Loop Control/Servo Gain 3-46

3.3 Velocity Control for Stepper Motors 3-47
3.3.1 Knee–Shaped Acceleration Characteristic 3-47
3.3.2 Parameterization of the Stepper Motor Frequency 3-49

3.4 Data Description 3-51

4 Manual Traversing and Handwheel Traversing 4-57

4.1 General Properties of Manual Traversing in JOG Mode 4-58

4.2 Controlling Manual Traversing via the PLC Interface 4-59

4.3 Continuous Traversing 4-60

4.4 Incremental Traversing (INC) 4-62

4.5 Handwheel Traversing in JOG Mode 4-63

4.6 Special Features of Handwheel Traversing 4-66
4.6.1 Monitoring Functions 4-66
4.6.2 Miscellaneous 4-67

4.7 Data Description 4-68

4.8 Signal Description 4-74
4.8.1 Overview of Signals to Axis/Spindle (Machine Axis) 4-78
4.8.2 Description of Signals to Axis/Spindle (Machine Axis) 4-78
4.8.3 Overview of Signals from Axis/Spindle (Machine Axis) 4-80
4.8.4 Description of Signals from Axis/Spindle (Machine Axis) 4-80

5 Program Mode 5-83

5.1 Operating Modes 5-84
5.1.1 Mode Change 5-85
5.1.2 Possible Functions in the Individual Modes 5-86
5.1.3 Monitoring Functions in the Individual Operating Modes 5-87
5.1.4 Interlocks in the Individual Modes 5-88

5.2 Program Test 5-89
5.2.1 Program Execution Without Axis Movements (Program Test) 5-89
5.2.2 Program Execution in Single Block Mode 5-90
5.2.3 Program Execution With Dry Run Feed 5-91

5.3 Processing of Certain Program Parts 5-92
5.3.1 Skipping Certain Part Program Blocks 5-93

5.4 Executing a Part Program 5-94
5.4.1 Part Program Selection 5-94
5.4.2 Starting the Part Program or Part Program Block 5-94
5.4.3 Interrupting a Part Program 5-95
5.4.4 RESET Command 5-96
5.4.5 Program Control 5-97
5.4.6 Program Status 5-98
5.4.7 Channel Status 5-99

5.5 Data Description 5-100

5.6 Signal Descriptions 5-102

6 Compensation 6-115

6.1 Backlash Compensation 6-116

6.2 Lead Error and Measuring System Error Compensation (LEC) 6-118

6.3 Drift Compensation 6-121

6.4 Data Descriptions 6-123

7 Face Axis 7-125

7.1 Radius / Diameter Programming: G22, G23 7-126

7.2 Constant Cutting Speed: G96 7-127

8 Reference–Point Approach 8-129

8.1 Fundamentals 8-129

8.2 Referencing Axes 8-131

8.3 Data Descriptions 8-135

9 Spindle 9-145

9.1 Spindle Modes 9-146
9.1.1 Spindle Control Mode 9-147
9.1.2 Spindle Positioning Mode 9-148
9.1.3 Spindle Positioning Mode 9-154

9.2 Referencing/Synchronizing 9-157

9.3 Speed and Gear Stage Change 9-158

9.4 Programming 9-163

9.5 Spindle Monitoring 9-164
9.5.1 Axis/Spindle on Standstill (n<nmin) 9-165
9.5.2 Spindle in Set Range 9-165
9.5.3 Max. Spindle Speed 9-165
9.5.4 Min./Max. Speed of Gear Stage 9-166
9.5.5 Max. Encoder Limit Frequency 9-167
9.5.6 Target Position Monitoring 9-168

9.6 Data Description 9-169

9.7 Signal Description 9-178

10 Output of Auxiliary Functions to PLC 10-187

10.1 Auxiliary Function Groups 10-188

10.2 Behavior with Block Search 10-188

10.3 Description of Auxiliary Functions 10-189

10.4 Data Description 10-190

10.5 Signal Description 10-192

11 Feeds 11-193

11.1 Overview 11-193

11.2 Feed F 11-194

11.3 Feed for Thread Cutting G33 11-195

11.4 Feed for Tapping with Compensating Chuck G63 11-196

11.5 Feed for Tapping without Compensating Chuck G331, G332 11-196

11.6 Rapid Traverse G0 11-197

11.7 Dry Run Feed 11-198

11.8 Velocity for Manual Traversing 11-199

11.9 Feed Override 11-200
11.9.1 Feed Lock and Feed/Spindle Stop 11-201
11.9.2 Feed Override from Machine Control Panel 11-202

11.10 Data Description 11-204

11.11 Signal Descriptions 11-204
11.11.1 Signals to Channel 11-205
11.11.2 Signals to Axis/Spindle 11-209

12 Tool Compensation 12-211

12.1 Tool 12-212

12.2 Data Description 12-214

13 EMERGENCY STOP 13-215

13.1 General 13-216

13.2 EMERGENCY STOP Tripping Mechanism 13-217

13.3 EMERGENCY STOP Sequence 13-217

13.4 EMERGENCY STOP Acknowledgement 13-219

13.5 Data Description 13-220

13.6 Signal Description 13-220

14 Various Interface Signals 14-223

14.1 General 14-224

14.2 Signals from PLC to NCK 14-226
14.3 Signals from NCK to PLC 14-230
14.4 Signals from PLC to MMC 14-232
15 List of Interface Signals 15-233
15.1 Interface Signals 15-234

Axis/Spindle Monitoring

1

Brief description

Any modern CNC must be provided with comprehensive monitoring mechanisms to protect man and machine.

The monitoring functions available include the following:

- Motion monitoring functions
  - Contour monitoring
  - Positioning monitoring
  - Zero–speed control
  - Clamping monitoring
  - Set speed monitoring
- Encoder monitoring functions
  - Encoder limit frequency
  - Zero mark monitoring
- Limit switch monitoring
- Stepper motor rotation monitoring

1.1 Motion Monitoring Functions

1.1.1 Contour Monitoring

Contour error

Contour errors are caused by signal distortions in the position control loop. A distinction is made between:

- Linear signal distortions. These are caused by:
  - Speed or position controller not being set optimally
  - Unequal servo gain factors for the feed axes involved in producing the contour

    If the servo gain factor of two axes involved in linear interpolation is equal, the actual point follows the setpoint on the same contour but with a delay.
    If the servo gain factors are not equal, a parallel offset between set contour and actual contour occurs.
  - Unequal dynamic response of the feed drives

    Unequal drive dynamics leads to contour deviations, in particular, at contour changes. Circles are distorted into ellipses due to unequal dynamics of the two feed drives.
- Non–linear signal distortions. These are caused by:
  - Activation of the current limitation within the machining range
  - Activation of the limitation of the set speed
  - Backlash within and / or outside the position control loop.

    Contour errors arise because of backlash and friction while travelling a circular contour.

    While travelling straight lines, a contour error occurs because of the backlash outside the position control loop, e.g. due to a tilting milling spindle. This causes a parallel offset between the actual and the set contour. The shallower the gradient of the straight line is, the larger is the offset.
  - Non–linear friction behavior of the slide guideways.

Effect

If the following error is too large, this has the following effect:

- Alarm 25050 ”Contour monitoring“ is output.
- The axis/spindle involved is stopped with rapid stop.

Rapid stop

The following applies to the spindle with SPOS motion and axes with analog drives: Stop (with open position control loop) via a speed setpoint ramp function.
The duration of the deceleration ramp is defined in MD:
AX_EMERGENCY_STOP_TIME (duration of the deceleration ramp for error states).
Stepper motor axes are stopped via an internal ramp.
If the axis is involved in interpolation with other axes, these are also stopped by rapid stop with clearing the following error (position partial setpoint = 0).

Remedy

- The following applies to the spindle and axes with analog drives:
  The real servo–gain factor must be set to correspond to the required servo gain factor set via MD: POSTCTRL_GAIN[N].
  Check MD: RATED_VELO (rated motor speed and
  MD: RATED_OUTVAL (rated output voltage).
- Check smooth running of axes/spindle.
- Check machine data for traversing motions
  (feed override, acceleration, max. velocities, ...)

1.1.2 Positioning Monitoring

Function

To ensure that an axis is positioned within a given time, the time configured in MD: POSITIONING:_TIME (time delay exact stop fine) is started on completion of the motion block (position partial setpoint=0 at the end of movement) and once this time has elapsed, a check is made to see whether the following error is below the limit value for STOP_LIMIT_FINE (Exact Stop Fine).

Effect

If the limit value for Exact Stop Fine is not yet reached when the positioning monitoring time has elapsed, the following action is performed:

- Alarm 25080 ”Positioning monitoring” is output.
- The axis/spindle concerned is stopped with rapid stop (see Section 1.1.1).

Error cause/ remedy

- Position controller gain too small --> change machine data for position controller gain
  MD: POSCTRL_GAIN[n] (servo gain factor)
- Positioning window (exact stop fine), positioning monitoring time
  and servo gain are not matched
  --> change machine data: MD: STOP_LIMIT_FINE (exact stop fine), MD: POSITIONING_TIME (delay time exact stop fine),
  MD: POSCTRL_GAIN[n] (servo gain factor)

Note

The size of the positioning window effects the block change time. The smaller these tolerances are selected, the longer the positioning task will take which in turn means a longer time before the next command can be executed.

1.1.3 Zero–Speed Control

Function

The zero–speed control has the following functionality:

- On completion of a motion block (position partial value=0 at the end of the motion), a check is made to see whether the following error has reached the limit for MD: STANDSTILL_DELAY_TIME (zero speed tolerance) after a parameterizable delay set in MD: STANDSTILL_POS_TOL (delay time zero speed control).
- On completion of the positioning process (axis stop fine reached), the zero speed control function takes over from the position monitoring function. A check is made to see whether the axis is moving more than specified in MD: STANDSTILL_POS_TOL (zero speed tolerance) from its position. Zero speed control is activated when ”Exact stop fine“ is reached and the ”Delay zero speed control set in MD: STANDSTILL_DELAY_TIME is still running.

Effect

When the monitoring function responds, it has the following effects:

- Output of alarm 25040 ”Zero speed monitoring“
- The axis/spindle involved is stopped with rapid stop (see Section 1.1.1).

Error cause/remedy

- Servo gain too large (oscillation of the control loop) --> change machine data for servo gain with axes equipped with analog drives or spindle:
  MD: POSCTRL_GAIN[n] (servo gain factor)
- Zero speed window too small --> change machine data
  MD: STANDSTILL_POS_TOL (zero speed tolerance)
- Axis is mechanically pushed out of position --> remedy cause

1.1.4 Clamping Monitoring

Function

If the axis is to be clamped on completion of the positioning process, clamping monitoring can be activated by means of the IS (interface signal) ”Clamping active“ (V380x0002.3).

This might be necessary because during the clamping process the axis can be pushed further out of the set position than the zero speed tolerance. The amount by which the set position is left is specified in MD: CLAMP_POS_TOL (clamping tolerance for interface signal clamping active).

Effect

If the axis is pushed out of position beyond the clamping tolerance during clamping, the following occurs:

- Output of alarm 26000 ”Clamping monitoring“
- The axis/spindle involved is stopped with rapid stop (see Section 1.1.1).

1.1.5 Set–Speed Monitoring

Function

Set–speed monitoring is used to check whether the physical limit of the spindle and axes equipped with analog drives (10V maximum voltage for set speed) is exceeded.

Effect

If the maximum set speed is exceeded, the following occurs:

- Output of alarm 25060 ”Set speed limitation“
- The axis/spindle is stopped with rapid stop (see Section 1.1.1).

Error cause/remedy

- Tacho compensation has not been carried out correctly or there is a measuring circuit or drive error.
- Setpoint too large (accelerations, velocities)

1.1.6 Actual Velocity Monitoring

Function

This is used to monitor the actual velocity for exceeding an admissible limit value entered in MD: AX_VELO_LIMIT[n] (threshold value for velocity monitoring).

Activation

The actual velocity monitoring function is always active if the measuring circuit activated by IS “Position measuring system” (V380x0001.5) provides actual values and is therefore still below the limit value.

It is active with:

- axes equipped with analog drives and stepper motor (the used stepper motors have no real measuring system)
- open–loop controlled and position–controlled spindles

Effect

If the “Threshold for velocity monitoring” is exceeded, the following occurs:

- Output of alarm 25030 ”Actual velocity alarm limit“
- The axis/spindle involved is stopped with rapid stop (see Section 1.1.1).

Remedy

- Check set–speed cable.
- Check actual values.
- Check direction of rotation.
- Check MD: AX_VELO_LIMIT[n] (threshold for velocity monitoring).

1.2 Encoder Monitoring Functions

1.2.1 Encoder Limit Frequency Monitoring

Function

(not applicable to stepper motor axes without encoder)
If the permissible limit frequency of a measuring system which is entered in MD: ENC_FREQ_LIMIT [0] (encoder limit frequency) is exceeded, the synchronization of the position (reference point) between machine and control system is lost. Correct position control is no longer possible.
This state is signaled to the PLC.

Activation

The encoder limit frequency monitoring function is always active when the encoder is switched on.

Effect

When the limit frequency of an encoder is exceeded, the following occurs:

- The IS ”Encoder limit frequency exceeded 1” (V390x0000.2) is set.
- The spindle continues running with speed control.
  If the spindle speed is reduced so much that the lower encoder limit frequency is violated, the spindle automatically synchronizes itself with the reference system of the measuring encoder again.
- If the limit frequency of the measuring system of a position–controlled axis is active, alarm 21610 ”Frequency exceeded“ is output.
- The axis/spindle involved is stopped with rapid stop (see Section 1.1.1).

Remedy

- Check encoder and encoder data.

1.2.2 Zero Mark Monitoring

Function

(not applicable to stepper motor axes)
Zero mark monitoring is used to check whether pulses have been lost between two zero mark passages of the actual position value encoder. The number of detected zero mark errors after which the monitoring function must respond is entered in MD: ENC_ZERO_MONITORING[0] (zero mark monitoring).

Activation

The monitoring function is activated with MD: ENC_ZERO_MONITORING[0].
Every time the encoder is switched on, counting of zero mark errors starts at ”0”.

Effect

If the number of zero mark errors entered in MD: ENC_ZERO_MONITORING[0]

- is reached while the measuring system is active, alarm 25020 ”Zero mark monitoring“ is output.

The spindle is stopped with rapid stop (see Section 1.1.1).

Error causes/remedy

- MD: ENC_FREQ_LIMIT [0] (encoder limit frequency) set too high.
- Encoder cable damaged.
- Encoder or encoder electronics defective.

1.2.3 Hardware Errors

Function

(not applicable to stepper motor axes)
In case of errors, the measuring circuit monitoring functions result in alarm 25000, 25001 ”Hardware error“.

Note

In case of hardware errors in the measuring circuit, the IS ”Referenced/Synchronized” (V390x0000.4) is canceled; i.e. the axis/spindle must be re–referenced/resynchronized.

1.3 Limit Switch Monitoring

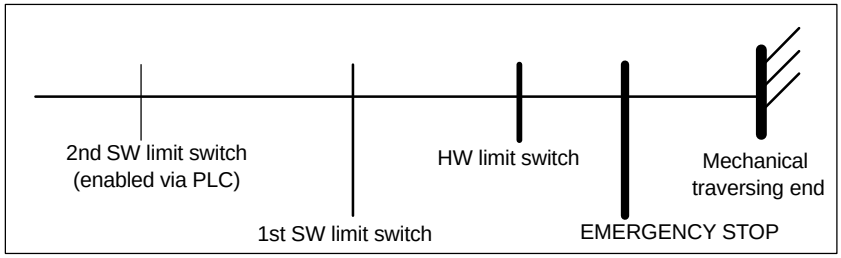


Fig. 1-1 Limit switches using the example of positive axis direction

Hardware limit switches

Function

For each axis there is a hardware limit switch for each direction to prevent the slide from being pushed out of the slide bed.
If the hardware limit switch is overshot, the PLC signals this to the NC via the IS “Hardware limit switch plus/minus” (V380x1000.1 /.0) and the motion of all axes is stopped.

Activation

HW limit switch monitoring is active in all operating modes after the control system has powered up.

Effect

- When a hardware switch is overshot in either direction, alarm 21614 ”Hardware limit switch + or -” is triggered.
- The direction keys in approach direction are disabled.

Remedy

- Move in the opposite direction (in JOG mode)
- Correct the program.

Software limit switches

Function

The software limit switches are intended to delimit the maximum treasuring range of each individual axis in normal mode.
2 software limit switch pairs are provided for each machine axis, which are defined via MD: POS_LIMIT_PLUS, POS_LIMIT_MINUS, POS_LIMIT_PLUS2, POS_LIMIT_MINUS2 (1st or 2nd software limit switch plus/minus) in the machine axis system.

Activation

- Software limit switch monitoring is active in all modes after reference–point approach.
- The position of the software limit switches can be approached.
- The 2nd software limit switch can be activated via the interface signal “2nd software limit switch plus/minus” (V380x1000.3 / .2) from the PLC in order to reduce the working area, for example, if a tailstock is swinged into position. The change becomes active immediately. The first software limit switch plus/minus is then deactivated.

Responses

The following responses are possible in each mode:

- If during the preparation of a block it is found that the axis position to be approached is greater than the positive/negative software limit switch, one of the following alarms is output:

  10720 ”Software limit switch + or -”
  10620 ”Axis reaches software limit switch + or -”

- If the position of a software limit switch is reached in JOG mode and you wish to travel further in this direction, alarm 10621 ”Axis reached software limit switch + or -” is output.
- If the monitoring function responds, the axis is decelerated with axis acceleration.
  If an axis is involved in interpolation with other axes, these axes will also be decelerated. This can lead tocontour violation.
- The execution of the program is aborted.
- The direction keys in approach direction are disabled.

Remedy

- Move in the opposite direction (in JOG mode).
- Correct the program.

1.4 Axis Monitoring Supplementary Conditions

To ensure that the monitoring functions operate correctly, particular attention has to be paid to the machine data:

– MD: LEADSCREW_PITCH (lead screw pitch)

– Gear ratio (load gear, encoder)
  MD: DRIVE_AX_RATIO_DENOM[n] (load gear denominator)
  MD: DRIVE_AX_RATIO_NUMERA[n] load gear numerator)
  MD: DRIVE_ENC_RATIO_DENOM[n] (measuring gear denominator)
  MD: DRIVE_ENC_RATIO_NUMERA[n] (measuring gear numerator)

– Motor speed / output voltage
  (applies to analog drives/spindle only
  MD: RATED_VELO (rated motor speed)
  MD: RATED_OUTVAL (rated output voltage)

– Encoder resolution

1.5 Stepper Motor Rotation Monitoring Using BERO

Overview

The BERO (proximity switch) for rotation monitoring is connected in the same way as with referencing with BERO.

Parallel connection with the BERO for referencing or using the same for rotation monitoring is possible. However, during referencing, either the rotation monitoring must be disabled or no switching edge may be provided from the reference BERO when rotation monitoring is active.

Modulo counter

A modulo conter (1 modulo = 1 revolution) is provided to count the actual–value increments. The modulo count is stored as a machine data.

MD: BERO_CYCLE	Repetition cycle of the BERO edges in actual–value increments

Activation

The rotation monitoring can be enabled/disabled via the IS ”Rotation monitoring” (V380x5000.0). When the BERO is overtraveled for the first time, the module count is stored as the BERO zeroing value to zero the modulo counter.

Comparison

With each further overtraveling of the BERO it is checked whether the contents of the modulo counter has its value in the vicinity of the stored BERO zero value.

A BERO tolerance can be taken into account via MD: BERO_EDGE_TOL. If the comparison yields a negative result, the IS ”Error: Rotation monitoring” is signaled to the PLC (V390x5000.0). The signal provides for edge evaluation signal and is only present as long as the PLC clock is provided. At the same time, the monitoring is automatically disabled, and rereferencing is required.

Note

The “Error: Rotation monitoring” occurs whenever the stepper motor is incorrectly controlled even if the rotation monitoring is not enabled. The user must make sure that the drive is switched off reliably. “Error: Rotation monitoring” means: Drive off!

1.6 Data Description

Machine data

31100 MD number	BERO_CYCLE[n] Steps between two BERO edges for rotation monitoring of the stepper motor		
Default: 2000	Min. input limit: 10	Max. input limit: 10 000 000	
Change effective after POWER ON	Protection level: 2/7	Unit: steps	
Data type: DWORD	Valid as from SW version:		
Meaning:	For the rotation monitoring of the stepper motor The number of steps between two equal BERO cycles must be entered		

31110 MD number	BERO_EDGE_TOL[n] Tolerance of BERO edges for rotation monitoring of the stepper motor		
Default: 50	Min. input limit: 10	Max. input limit: 10 000 000	
Change valid after POWER ON	Protection level: 2/7	Unit: steps	
Data type: DWORD	Valid as from SW version:		
Meaning:	The number of steps of the stepper motor must be entered in order to tolerate the BERO edges occurring during the rotation monitoring.		

31350 MD number	FREQ_STEP_LIMIT Maximum stepper motor frequency		
Default: 250000.0	Min. input limit: 100	Max. input limit: 1 000 000	
Change valid after NEW_CONF	Protection level: 2/7	Unit: Hz	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	MD comes into effect with stepper motor drive; maximum frequency that may occur with a stepper motor		

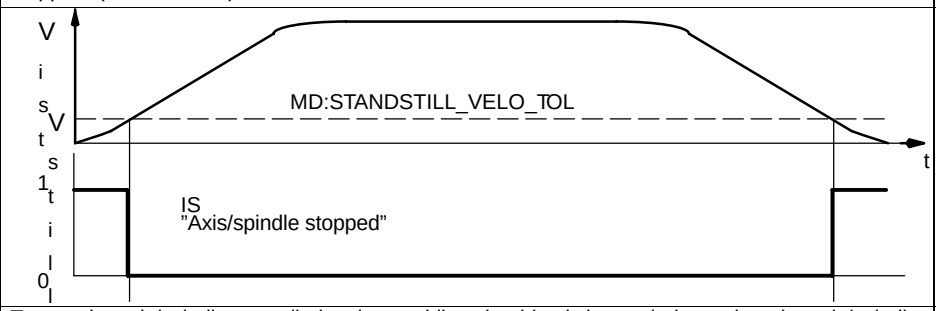
36020 MD number	POSITIONING_TIME Delay Exact stop fine		
Default: 5	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The time after which the following error must have reached the limit value for Exact Stop Fine when the position is approached (position partial setpoint=0). If this is not the case, alarm 25080 ”Position monitoring“ is output and the axis concerned stopped. The MD should be selected such that the monitoring function in normal mode does not respond, since the entire traversing process (acceleration, constant traversing, deceleration) is monitored by other functions without gaps.		
Related to	MD: STOP_LIMIT_FINE (exact stop fine)		

36030 MD number	STANDSTILL_POS_TOL Zero–speed tolerance		
Default: 0.2	Min. input limit:: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: mm, degrees	
Data type: DOUBLE	Valid as from SW version:		
Related to	MD: STANDSTILL_DELAY_TIME (delay time zero speed control)		

36030 MD number	STANDSTILL_POS_TOL Zero–speed tolerance
Meaning:	This MD is used as a tolerance band for the following monitoring functions: • On completion of a motion block (position partial value=0 at the end of motion) it is monitored whether the following error has reached the limit value for STANDSTILL_POS_TOL (zero–speed tolerance) after the parameterizable STANDSTILL_DELAY_TIME (delay time after zero speed control). • On completion of a positioning process (exact stop fine reached), zero–speed monitoring takes over positioning monitoring. It is monitored whether the axis in MD: STANDSTILL_POS_TOL (zero–speed tolerance) moves from its position more than specified. If the actual position is below or across the set position by the amount of the zero–speed tolerance, alarm 25040 ”Zero–speed monitoring“ is output and the axis is stopped.
Special cases, errors, ...	The zero–speed tolerance must be greater than the Exact Stop Tolerance Range Coarse.
Related to	MD: STANDSTILL_DELAY_TIME (delay time zero speed control)

36040 MD number	STANDSTILL_DELAY_TIME Delay time zero–speed control		
Default: 0.2	Min. input limit:: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	See MD: STANDSTILL_POS_TOL (zero speed control)		
Related to	MD: STANDSTILL_POS_TOL (zero speed control)		

36050 MD number	CLAMP_POS_TOL Clamping tolerance at interface signal ”Clamping active“		
Default: 0.5	Min. input limit:: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: mm; degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The interface signal ”Clamping process active“ (V380x0002.3) activates clamping monitoring. If the axis under monitoring is pushed out of the set position (exact stop tolerance range) by an amount greater than the clamping tolerance, alarm 26000 ”Clamping monitoring” is generated and the axis stopped.		
Special cases, errors, ...	The clamping tolerance must be greater than the Exact Stop Tolerance Range Coarse.		
Related to	IS ”Clamping process active“		

36060 MD number	STANDSTILL_VELO_TOL Maximum velocity/speed ”Axis/spindle stopped“		
Default: 5	Min. inputn limit: 0	Max. input limit: plus	
Change valid after NEW_CONF	Protection level: 2/7	Unit: Linear axis: mm/min Spindle: rpm	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	This machine data defines the zero speed range for the axis velocity or spindle speed. If the current actual velocity of the axis or the actual speed of the spindle is less than the entered value and if no more setpoints are output from the NC to the axis/spindle, the IS “Axis/spindle stopped” (V390x0001.4) is set.		
			
Application example(s)	To stop the axis/spindle controlled, pulse enabling should only be carried out when the axis/spindle is at a standstill. Otherwise, the axis would coast to stop.		
Related to	IS ”Axis/spindle stopped“ (V390x0001.4)		

36100 MD number	POS_LIMIT_MINUS 1st software limit switch minus		
Default: - 100 000 000	Min. input limit:: ***	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: mm; degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Meaning as 1st software limit switch plus, but for the traversing range limit in the negative direction. The MD is effective after reference–point approach if the PLC interface signal “2nd software limit switch minus” is not set.		
MD not applicable if	the axis is not referenced		
Related to	IS ”2nd software limit switch minus“		

36100 MD number	POS_LIMIT_PLUS 1st software limit switch plus		
Default: 100 000 000	Min. input limit:: ***	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: mm; degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In addition to the hardware switch, it is also possible to use a software limit switch. The absolute position in the machine axis system of the positive range limit of each axis is entered. The MD is effective after reference–point approach if the IS “2nd software limit switch plus” is not set.		
MD not applicable if.....	the axis is not referenced		
Related to	IS ”2nd software limit switch plus“		

36120 MD number	POS_LIMIT_MINUS2 2nd software limit switch minus		
Default: - 100 000 000	Min. input limit:: ***	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: mm; degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Meaning as 2nd software limit switch plus, but for the traversing range limit in the negative direction. Which of the two software limit switches 1 or 2 is to be activated, can be selected by an interface signal from the PLC. e.g. V38011000.2 = 0 ”1st software limit switch minus” for the 1st axis active V38011000.2 = 1 ”2nd software limit switch minus“ for the 1st axis active		
MD not applicable if	the axis is not referenced		
Related to	IS ”2nd software limit switch minus”		

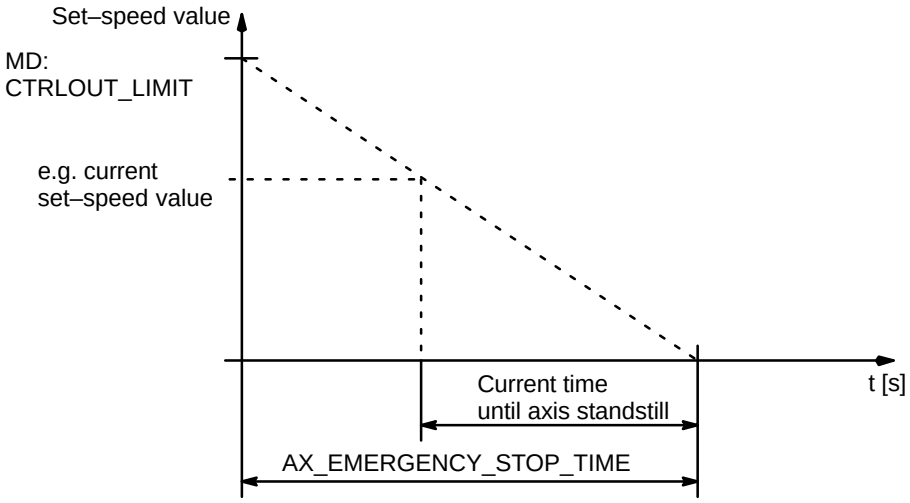
36130 MD number	POS_LIMIT_PLUS2 2nd software limit switch plus		
Default: 100 000 000	Min. input limit:: ***	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: mm; degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	This machine data can be used to specify a second SW limit switch position in positive direction in the machine axis system. Which of the two SW limit switches 1 or 2 is to be active, can be selected from the PLC by means of an interface signal. e.g.: V38011000 bit 3 = 0 ”1st software limit switch plus” for 1st axis active V38011000 bit 3 = 1 ”2nd software limit switch plus” for 1st axis active		
MD not applicable if	axis is not referenced.		
Related to	IS ”2nd software limit switch plus“		

36200 MD number	AX_VELO_LIMIT[n] Threshold value for velocity monitoring		
Default: 11500	Min. input limit:: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: mm/min rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The threshold value of actual velocity monitoring is entered in this MD. If the threshold value is exceeded, alarm 25030 ”Actual speed alarm limit” is output and the axes stopped. Settings: • For the axes, a value that is by 10 ... 15 % higher than set via MD: MAX_AX_VELO (maximum axis velocity) should be selected. The following should therefore be applicable to the threshold value of velocity monitoring: MD: AX_VELO_LIMIT[n] > MD: MAX_AX_VELO * (1,1 ... 1,15) • For the spindle, the value per gear stage should be selected by 10-15 % higher than set via MD: GEAR_STEP_MAX_VELO_LIMIT[n] (maximum speed of gear stage). The index [n] of the machine data is coded as follows: [Servo parameter block No.]: 0-5		

36300 MD number	ENC_FREQ_LIMIT[n] Encoder limit frequency		
Default: 300000	Min. input limit:: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: Hz	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The encoder limit frequency is entered in this machine data.		

36302 MD number	ENC_FREQ_LIMIT_LOW Encoder frequency for restart		
Default: 99,9	Min. input limit: 0	Max. input limit: 100	
Change valid after NEW_CONF	Protection level: 2/7	Unit: %	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The encoder frequency monitoring uses a hysteresis. ENC_FREQ_LIMIT defines the encoder limit frequency at which the encoder is turned off, and ENC_FREQ_LIMIT_LOW defines the frequency at which the encoder is turned on again. ENC_FREQ_LIMIT is specified directly in Hertz. ENC_FREQ_LIMIT_LOW, however, is a portion of ENC_FREQ_LIMIT specified as a percentage. Normally, the default of MA_ENC_FREQ_LIMIT_LOW is sufficient. When absolute encoders with EnDat interface are used, however, the limit frequency of the absolute track is considerably lower than the limit frequency of thre incremental track. Using a small value in ENC_FREQ_LIMIT_LOW, it can be achieved that the encoder is turned on only below the limit frequency of the absolute track and therefore only references if this is admitted by the absolute track. This referencing is done for spindles automatically.		

36310 MD number	ENC_ZERO_MONITORING[n] Zero mark monitoring		
Default: 0	Min. input limit:: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: -	
Data type: DWORD	Valid as from SW version:		
Meaning:	This machine data is intended to activate zero mark monitoring and to define the number of illegal zero mark errors. 0: no zero mark monitoring > 0: number of detected errors at which monitoring is to be responded.		
Examples:	ENC_ZERO_MONITORING[0] = 2 ⇒ 1st error is ignored; monitoring function responds with 2nd error. After the encoder has been turned on, the number of zero mark errors is reset to ”0”.		

36610 MD number	AX_EMERGENCY_STOP_TIME Duration of deceleration ramp during error conditions		
Default: 0.05	Min. input limit:: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The spindle is stopped with rapid stop (open position control loop) via a deceleration ramp of the set–speed value if the following monitoring functions respond: • Positioning monitoring • Zero–speed control • Clamping monitoring • Set–speed value monitoring • Actual–speed value monitoring • Encoder limit frequency monitoring (except for speed–controlled spindle) • Zero mark monitoring The duration for the reduction of the set–speed value from the maximum set–speed value to the set value = 0 must be entered in the MD. The duration until standstill depends on the current set–speed value when a monitoring function responds.  Fig. 4-1 Deceleration ramp during error conditions		
Meaning:	With interpolating axes, compliance with the contour during a deceleration phase is not guaranteed. CAUTION: If the duration of the deceleration ramp during error conditions is set too large, servo enable will be canceled although the spindle is still moving. It is then rapidly stopped with set–speed value 0. For this reason, the time in MD: AX_EMERGENCY_STOP_TIME should be less than the time in MD: SERVO_DISABLE_DELAY_TIME (servo enable switch–off delay).		
Related to	MD: SERVO_DISABLE_DELAY_TIME Servo enable switch–off delay MD: CTRLOUT_LIMIT Maximum set–speed value		

1.7 Signal Description

Signals to axis/spindle

V380x0002.3 Interface signal	Clamping process active Signal(s) to axis/spindle (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Clamping process active Clamping monitoring is activated.	
Signal status 0 or edge change 1 ---> 0	Clamping process completed. Clamping monitoring is taken over by zero speed control.	
Related to	MD: CLAMP_POS_TOL (clamping tolerance)	

V380x0003.6 Interface signal	Velocity/spindle speed limitation Signal(s) to axis/spindle (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The NCK limits the velocity/spindle speed to the limit value entered in MD: SPIND_EXTERN_VELO_LIMIT.	
Signal staus 0 or edge change 1 ---> 0	No limitation active.	
Related to	MD: SPIND_VELO_LIMIT (max. spindle speed) MD: SPIND_MAX_VELO_G26 (programmed spindle speed limitation G26) MD: SPIND_MAX_VELO_LIMS (programmed spindle speed limitation G96)	

V380x1000.1 and./0 Interface signal	Hardware limit switch plus and minus Signal(s) to axis/spindle (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	On both ends of the traversing range, one switch each can be installed, which provides a Hardware Limit Switch Plus or Minus signal to the NC during approach via the PLC. When the signal is detected as set, alarm 021614 ”Hardware limit switch + or -” is output and the axis is decelerated immediately. The deceleration method is set by means of MD: deceleration_MODE_CHOICE (deceleration response with hardware limit switch). If servo enable for the Hardware Limit Switch signal is canceled, the axis will react as explained in the Functional Description, “Diverse Interface Signals A2”.	
Signal status 0 or edge change 1 ---> 0	Normal status, no HW switch responded.	
Related to		

V380x1000.3 and .2 Interface signal	2nd software limit switch plus or minus Signal(s) to axis/spindle (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	2nd software limit switch for plus or minus direction is active. 1st software limit switch for plus or minus direction is inactive. In addition to the 1st software limit (plus or minus), the 2nd software limit switches (plus or minus) can be activated via these interface signals. The position is defined via the MD: POS_LIMIT_PLUS2, POS_LIMIT_MINUS2 (2nd software limit switch plus, 2nd software limit switch minus).	
Signal status 0 or edge change 1 ---> 0	1st software limit switch for plus or minus direction is active. 2nd software limit switch for plus or minus direction is inactive.	
Related to	MD: POS_LIMIT_PLUS, POS_LIMIT_PLUS2, POS_LIMIT_MINUS, POS_LIMIT_MINUS2, (software limit switch plus, software limit switch minus)	

V380x5000.0 Interface signal	Rotation monitoring Signal(s) to axis (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge evaluation 0 → 1	Rotation monitoring active (further information – see Chapter LEERER MERKER)	
Signal status 0 or edge change 1 → 0	Rotation monitoring OFF	
Related to	IS "Error: Rotation monitoring" (V390x5000.0)	

Signals from axis/spindle

V390x0000.2 Interface signal	Encoder limit frequency exceeded 1 Signal(s) from axis/spindle (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge evaluation 0 ---> 1	The limit frequency set in MD: ENC_FREQ_LIMIT(encoder limit frequency) is exceeded. The reference point /the synchronization for the position–measuring system is lost (IS: Referenced/Synchronized has signal status 0). Position control is no longer possible. The spindle continues turning with speed control.	
Signal status 0 or edge evaluation 1 ---> 0	The limit frequency set in MD: ENC_FREQ_LIMIT is not exceeded.	

V390x5000.0 Interface signal	Error Rotation monitoring Signal(s) from axis/spindle (NCK → PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge evaluation 0 → 1	Error in rotation monitoring of this stepper motor axis (further information – see Chapter 14.3)	
Signal status 0 or edge change 1 → 0	No error in rotation monitoring of this stepper motor axis	
Related to	IS "Rotation monitoring ON/OFF" (V380x5000.0)	

Continuous–Path Control, Exact Stop

2

Brief description

For continuous–path control, the CNC executes a part program successively block by block. The next block is only executed if the functions of the currently active block are executed. Different requirements made to the part to be machined, such as contour accuracy, machining time, workpiece surface, require different block change criteria. Two behavior types are possible for the contour axes at the block ends. The first type, exact stop, means that all contour axes must have reached the set target position depending on an exact stop criterion before the next block change is initiated. In order to be able to fulfil this criterion, the contour axes must reduce the contour velocity with each block change what, however, means a delay of the block change. The second type, continuous–path control mode, is a try to avoid deceleration of the contour velocity in order to change to the next block with possibly the same contour velocity.

The following function description describes the features and possibilities of the Exact Stop function and the Continuous–Path Control Mode.

2.1 General

Contour axes

Contour axes are all machining axes guided by an interpolator, which determines the contour points, such that

- all axes involved start at the same time;
- each of the axes involved traverse with the correct velocity ratio;
- all axes reach the programmed target position at the same time.

Depending on the particular contour, e.g. circle, the accelerations of the individual axes can be different.

2.1.1 Velocities

Contour axes are subject to the axis–specific velocity limit values and acceleration limit values.

Feed

The programmed feed F corresponds to the feedrate. It is modal and is programmed as a velocity, e.g. in the units mm/min (or inch/min) with G94, or in mm/rev. (or inch/rev.) with G95. The feed represents the geometric total of the feedrates of the axes involved in the interpolation. The feed is specified for the movement types G1, G2, G3 and G5. If the contour axes for these movements are programmed without feed, alarm 10860 “No feed programmed” is output.

Feed override

The feed override is effective for all contour axes together.

Rapid traverse

Rapid traverse G0 is the function with which the maximum contour velocity can be reached. When an axis is traversed with rapid traverse, the maximum axis velocity of the axis limits the rapid traverse speed.

Rapid traverse override

The rapid traverse override is effective for all axes together.

Velocity for zero–cycle blocks

Zero–cycle blocks are blocks whose path length is shorter than the path which can be traversed using the programmed set feed and the interpolator cycle. For accuracy reasons, the velocity is reduced such that at least one interpolator cycle is required for the path. The velocity is thus equal to or less than the quotient of the block path length divided by the IPO cycle.

2.1.2 Stopping for Synchronization

Irrespective of whether exact stop or continuous–path control mode is selected, the block change can be delayed by synchronization processes and thus cause the contour axes to stop. In exact stop mode, the contour axes are stopped at the end point of the current block. In the same situation, in continuous–path control mode, the axes are stopped at the next block end point at which they can be decelerated with compliance of their acceleration limits. They are stopped for synchronization

- in case of PLC acknowledgement.
  If acknowledgement by the PLC is required for an auxiliary function which is output prior to or after the end of a movement, the axes are stopped at the end of the block.
- if the subsequent blocks are not provided.
  If the subsequent blocks cannot sufficiently fast be made available for execution, the axes are stopped at the last approachable block end.
- when the buffer is cleared.
  If the NC program requires that the advance can be synchronized with the main run (e.g. by means of the STOPRE instruction – Clear buffer), this implies a block–related velocity reduction or exact stop.

When the axes are stopped for synchronization, no contour errors occur. However, in particular, in continuous–path control mode, stopping is not desired because relief cutting can occur.

2.2 Exact Stop

The Exact Stop function is used to wait for the contour axes running into the programmed block end point. When all axes have reached the exact stop criterion, the block change is carried out. The velocity at the block transition is approximately zero.

This means:

- that all contour axes reach nearly standstill in the block end point without overshooting.
- The machining time is extended due to the waiting time for reaching the exact stop criterion.
- Relief–cutting can occur due to the waiting time for reaching the exact stop criterion.

The exact stop function is suited for exact traversing of contours.

Exact stop is not recommended if

- the exact contour within a criterion (e.g. exact stop fine) may differ from the programmed contour in order to achieve faster machining.
- absolute velocity tolerance is required.

Exact stop activation

The exact stop function can be selected in the NC program either by means of the command G60 or G09. The desired exact stop criterion should be specified with the associated program code prior to or with the selection. G60 is modal, and G09 is non–modal. G09 is used to interrupt the continuous–path control mode. Both exact stop functions will only be active with the selected exact stop criterion. The exact stop function is deselected with continuous–path control mode.

Exact stop criteria

- •
  Exact stop fine
  This criterion is used to monitor whether the actual position of the axis is away from the set position within a certain path distance. The size of the per permitted distance is stored in MD: STOP_ LIMIT_FINE (exact stop fine).
- Exact stop coarse
  The same scope of functions as exact stop fine, but the monitoring window is defined in MD: STOP_LIMIT_COARSE (exact stop coarse). In order to achieve a faster block change as with the exact stop fine criterion, the exact stop coarse window must be parametrized larger than the exact stop fine window.

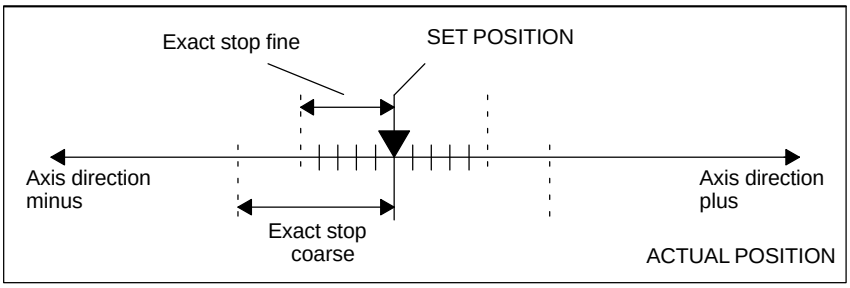


Fig. 2-1 Representation of the exact stop tolerance ranges fine/stop

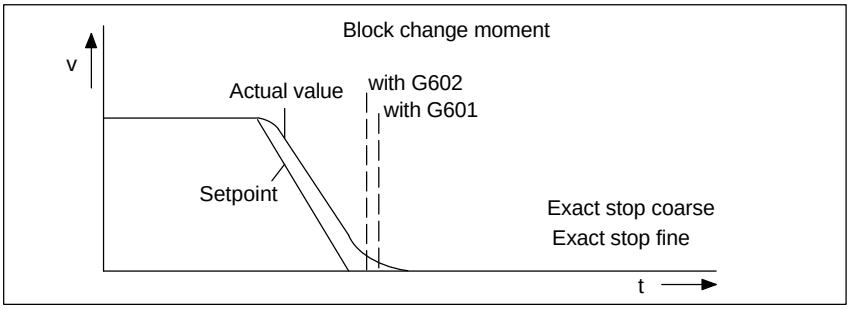


Fig. 2-2 Block change depending on the exact stop criteria

Exact stop criteria activation

The exact stop criteria can be selected in each NC part program block by means of the G codes

- G601 –exact stop fine
- G602 – exact stop coarse

and are evaluated by means of the exact stop functions G60 or G09. An active criterion is deactivated by selecting another criterion.
Certain situations can imply exact stop in continuous–path control, which will then consider one of the three criteria (see also ”implicit exact stop“ in Section 2.3).

2.3 Continuous–Path Control Mode

In continuous–path control mode, the contour velocity for block change at the end of the block is not decelerated to a velocity which allows the exact stop criterion to be reached. The aim is to avoid higher axis deceleration of the contour axes at the block change moment in order to be able to change to the next block with possible the same contour velocity.

Continuous–path control mode is used to smooth bent block transitions by local modifications to the programmed curve or to make them tangentially.

Continuous–path control has the following effects:

- Rounding of the contour corners.
- Shorter machining times thanks to missing deceleration and acceleration processes required to reach the exact stop criterion.
- Better cutting conditions due to more even velocity curve.

Continuous–path control mode is recommended whenever :

- a contour is to be traversed as fast as possible;
- the exact curve within an error criterion may deviate from the programmed curve in order to generate a continuous curve over the entire range.

Implicit exact stop

In some cases, it is necessary to generate exact stop in continuous–path control mode in order to be able to execute consequential reactions. In these situations, the contour velocity is reduced to zero.

- If auxiliary functions are output prior to the traversing movement, the preceding block is only completed when the selected exact stop criterion is reached.
- If auxiliary functions are output after the traversing movement, these are output after the interpolator end of the block.
- If the function “Clear buffer” is programmed in the part program, the preceding block is completed when the selected exact stop criterion is reached.

Velocity = 0 in continuous–path control mode

Irrespective of the implicit exact stop, the contour motion at the end of the block is decelerated to zero velocity if:

- the time required to position a spindle which has been programmed with the syntax SPOS is longer than the traversing time of the contour axes. The block change is carried out when exact stop fine of the positioning spindle is reached.
- stopping for synchronization is required.

Auxiliary function output during traverse

In continuous–path control mode with auxiliary function output and **short traversing blocks**, the contour velocity is decelerated already prior to the acknowledgement by the PLC. The axes are thus stopped at the end of the block in compliance with the acceleration limits. Acknowledgement is waited there in order to continue the movement.

2.4 Data Description

Machine data

36000 MD number	STOP_LIMIT_COARSE Exact stop coarse		
Default: 0.04	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	User class: 2/7	Unit: mm, degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	An NC block is considered as completed if the actual position of the contour axes is away from the set position by the value of the entered exact stop tolerance range. If the actual position of a contour axis is not within this limit, the NC block is not considered completed and further part program execution is not possible. The change to the next block can be influenced by the size of the entered value. The greater the value is selected, the earlier the block change is initiated. If the set exact stop tolerance range is not reached – the block is not considered completed. – the axis cannot be traversed again. – alarm 25080 (positioning monitoring) is output after the time set in MD: POSITIONING_TIME (monitoring time exact stop fine) has elapsed. – the direction of movement +/– for the axis is displayed in the positioning display. The exact stop window is also evaluated for the spindle in position–controlled mode.		
Special cases, errors, ...	MD: STOP_LIMIT_COARSE may not be set to a value less than MD: STOP_LIMIT_FINE (exact stop fine). In order to achieve the same block change behavior as with the exact stop fine criterion, the exact stop coarse window may be equal to the exact stop fine window. The MD: STOP_LIMIT_COARSE may not be set to a value equal to or greater than the MD: STANDSTIL_POS_TOL (zero–speed tolerance).		
Related to	MD: POSITIONING_TIME (delay time exact stop fine)		

36010 MD number	STOP_LIMIT_FINE Exact stop fine		
Default: 0.01	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	User class: 2/7	Unit: mm, degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	See MD: STOP_LIMIT_COARSE (exact stop coarse)		
Special cases, errors, ...	The MD: STOP_LIMT_FINE may not be set to a value greater than MD: STOP_LIMIT_COARSE (exact stop coarse). The MD: STOP_LIMIT_FINE may not be set to a value equal to a greater than the MD: STANDSTILL_POS_TOL (zero–speed tolerance).		
Related to	MD: POSITIONING_TIME (delay time exact stop fine)		

36020 MD number	POSITIONING_TIME Delay time exact stop fine		
Default: 5.0	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	User class: 2/7	Unit: s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	If a block is ended with exact stop, the axis must have reached the exact stop fine/coarse window within the positioning time. Otherwise, the positioning process is aborted with alarm 25080 “Positioning monitoring“. The monitoring time is started with the interpolator end of the axis. A position–controlled spindle is also subject to this time–related positioning monitoring. In case of error, alarm 25080 “Positioning monitoring” is output. Alarm 25080 cancels the IS ”READY” (V31000000.3) and stops the axes/position–controlled spindle.		
Related to	MD: STOP_LIMIT_COARSE (exact stop coarse) MD: STOP_LIMIT_FINE (exact stop fine)		

2.5 Signal Description

V33000004.3 Interface signal	All axes have stopped Signal(s) from channel (NCK -> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	All axes and the position–controlled spindle stop with interpolator end. No further traversing movements will be carried out.	

V390x0000.6 Interface signal	Position reached with exact stop coarse Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	The axis is in the respective exact stop; the interpolator for the axis is no longer active (set position reached). Or, the interpolator is not active, since – the control system is in RESET condition (Reset button or end of program); – the contour movement has been ended with NC stop. – the spindle is in position–control mode (SPOS instruction) and is on standstill.	
Signal status 0	The axis is no longer in the respective exact stop.	
Related to	MD: STOP_LIMIT_COARSE (exact stop coarse)	

V390x0000.7 Interface signal	Position reached with exact stop fine Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW–version:
Signal status 1 or edge change 0 ——> 1	See IS ”Position reached with exact stop coarse“	
Signal status 0 or edge change 1 ——> 0	See IS ”Position reached with exact stop coarse“	
Related to	MD: STOP_LIMIT_FINE (exact stop fine)	

Velocity, Setpoint/Actual–Value System, Closed–Loop Control

3

Brief description

This Section describes the adjustment of

- the measuring systems
- the setpoint value system
- the positioning accuracy
- the traversing ranges and
- the axis velocities.

Velocities

The maximum path, axis velocity and spindle speed are affected by the machine and drive dynamics and the limit frequency for actual–value sensing (encoder limit frequency).

The maximum axis velocity is defined in machine data MAX_AX_VELO (maximum axis velocity). The maximum permissible spindle speed is set in MD: SPIND_VELO_LIMIT (maximum spindle speed).

In addition to the limitation by MD: MAX_AX_VELO, the control system limits the maximum path velocity on the situation with the following formula:

$$V_{max} \leq \frac{\textit{Progr. path length in a part program block[mm or degrees]}}{\textit{IPO cycle [s]}} * 0.9$$

With a higher feedrate (resulting from the programmed feedrate and the feed–rate override), the maximum path velocity is limited to $V_{max.}$

This automatic feedrate limitation can lead to a drop in velocity over several blocks with programs generated by CAD systems with extremely short blocks.

Example:
IPO cycle = 12 ms
N10 G0 X0 Z0; [mm]
N20 G0 X100 Z100; [mm]
⇒ programmed path length in block = 141.42 mm

⇒ V_{max} = (141.42 mm / 12 ms) * 0.9 = 10606.6 mm/s = 636.39 m/min

The following restriction applies to the minimum path or axis velocity:

$$V_{min} \geq \frac{10^{-3}}{\textit{Calculation resolution}[\frac{\textit{incr.}}{\textit{mm or degrees}}] * \textit{IPO cycle [s]}}$$

The calculation resolution amounts to 1,000 incr/mm or incr./degrees.

If the velocity drops below V_{min} **no** traverse movement takes place!

Example: IPO cycle = 12 ms;
⇒$V_{min} = 10^{-3}$ / (1000 $^{Incr}/_{mm}$ x 12 ms) = 0.005 ^{mm} / $_{min}$;

Value range forpath feedrate F:
Metric system:
0.001 ≤ F ≤ 999,999.999 [mm/min, mm/rev]
Inch system:
0.001 ≤ F ≤ 399,999.999 [inch/min, inch/rev]

Value range for spindle speed S:
0.001 ≤ S ≤ 999,999.999 [rev/min]

Traversing ranges

Table 3-1 Traversing ranges of the axes

	G71 [mm]	G70 [inch]
	Range	Range
Linear axes X, Z	± 999,999.999	± 399,999.999
Interpolation parameters I, J, K	± 999,999.999	± 399,999.999

The traversing range can be limited by software limit switches.

Positioning accuracy of the control

The positioning accuracy of the control system depends on the actual–value resolution (= encoder increments / (mm or degrees)) and on the calculation accuracy (= internal increments / (mm or degrees)).

The coarser resolution of the two values determines the positioning accuracy of the control system.

The choice of input resolution, interpolator and position control cycle have no effect on this accuracy.

Metric/inch measuring system, basic system

The control system can use inch or metric systems. The basic setting is defined in MD: SCALING_SYSTEM_IS_METRIC (basic system metric). Depending on the setting in the MD, all geometric values are interpreted either as metric or inch values. All manual settings also refer to this basic setting (e.g. handwheel, INC, feedrate), as do zero offsets, tool offsets, etc. and the associated displays.

Converting the basic system

In part programs, the workpiece–related specifications can be switched over between the measuring systems by means of G70/G71. The data affected by G70/G71 is described in the Programming Instructions

Standardization of physical quantities in the machine and setting data

Machine and setting data that contain physical quantities are interpreted as standard in the following input/output units depending on the basic system used (metric/inch):

Physical quantity:	**Input/output units for standard basic system:**	
	Metric	**Inch**
Linear position	1 mm	1 inch
Angular position	1 degree	1 degree
Linear velocity	1 mm/min	1 inch/min
Angular velocity	1 rev/min	1 rev/min
Linear acceleration	1 m/s^2	1 $inch/s^2$
Angular acceleration	1 rev/s^2	1 rev/s^2
Linear jerk	1 m/s^3	1 $inch/s^3$
Angular jerk	1 rev/s^3	1 rev/s^3
Time	1 s	1 s
Position controller loop gain	1/s	1/s
Revolutional feedrate	1 mm/rev	1 inch/rev
Compensation value Linear position	1 mm	1 inch
Compensation value Angular position	1 degree	1 degree

3.1 Setpoint/Actual–Value System

3.1.1 General

Block diagram

A control loop with the following structure can be configured for every closed–loop controlled axis/spindle (for the stepper motor axes the encoder must be considered as an internal encoder):

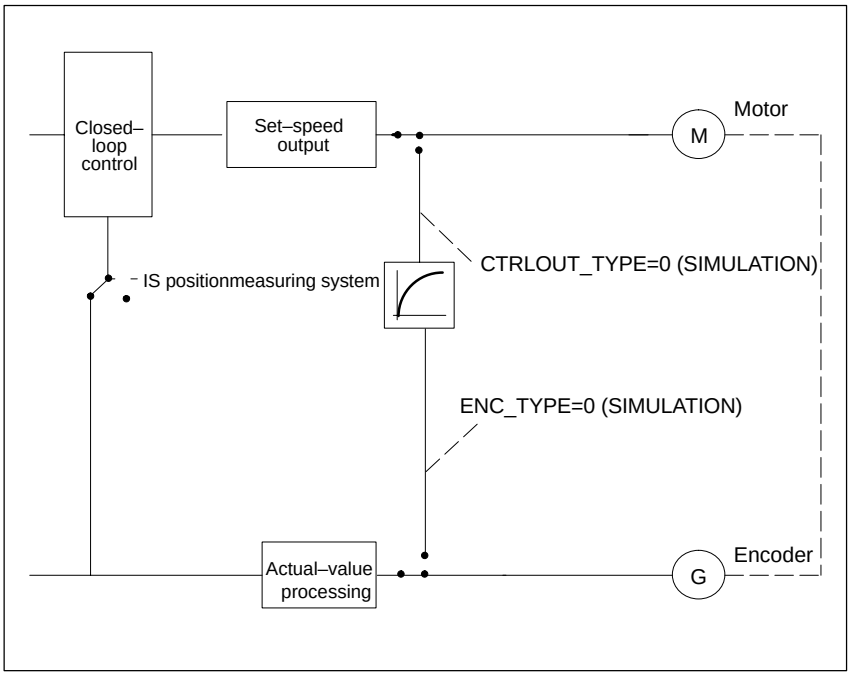


Fig. 3-1 Block diagram of a control loop

Setpoint output

One setpoint can be output for each axis/spindle. Setpoint output to the final control element is performed as an analog value for the spindle and axis with analog drive (format ± 10 V). For stepper motor axes, signal output is provided for pulse and direction (see “Start–Up Guide“, Section “Connecting the Feed Drives“).

Actual–value sensing

A square–wave generator (standard, increment quadrupling) can be connected to the spindle/ axis.

No encoder is required for the stepper motor axes.

Simulation axes

The speed control loop of an axis/spindle can be simulated for testing purposes. The axis “traverses” with a following error, similarly to a real axis.

A simulation axis is defined by setting the two
MD: CTRLOUT_TYPE[0] (setpoint output type) and ENC_TYPE[0] (actual–value sensing type) to ”0”.

As soon as the standard machine data are loaded, the axes become simulation axes.

Setpoint and actual value can be set to the reference–point value by reference–point approach.

It is also possible to define via the MD: SIMU_AX_VDI_OUTPUT (output of axis signals for simulation axes) whether the axis–specific IS are to be output to the PLC during simulation.

3.1.2 Speed Setpoint Output and Actual–Value Processing

Control direction and traversing direction of the feed axes

Control direction

The MD: ENC_FEEDBACK_POL[n] (sign of actual value) can be used to change the sign of actual–value sensing and thus the control direction of position control.

Traversing direction

The MD: AX_MOTION_DIR (traversing direction) can be used to reverse the direction of movement of the axis, without affecting the control direction of the position control.

Speed setpoint adjustment / tacho adjustment

General

The machine data RATED_VELO[n] determines the rated motor speed.

Basic setting

The MD: RATED_VELO determines the rated motor speed.
MD: RATED_OUTVAL [(rated output voltage) tells the control system which speed setpoint voltage corresponds to which motor speed (not with stepper motor axes).
In MD: RATED_OUTVAL (rated output voltage), enter the value of the speed setpoint at which the motor speed specified in RATED_VELO (rated motor speed) as a percentage with reference to the maximum speed setpoint.

If the motor speed is not known, it can be calculated from the desired axis velocity, the leadscrew pitch MD: LEADSCREW_PITCH and the gear ratio
MD: DRIVE_AX_RATIO_NUMERA[n] (load gear numerator)
MD: DRIVE_AX_RATIO_DENOM[n] (load gear denominator) as follows

$$n_{motor} = \frac{V_{axis} * R}{S}$$

where the following applies:
n_{motor} ≒ motor speed;
V_{axis} ≒ axis velocity;
S ≒ leadscrew pitch;
(MD: LEADSCREW_PITCH)
R ≒ gear ratio

where the following applies:

$$R = \frac{\text{Number of motor revolutions}}{\text{Number of load (spindle) revolutions}} = \frac{DRIVE_AX_RATIO_NUMERA[n]}{DRIVE_AX_RATIO_DENOM[n]};$$

Accuracy

The accuracy of this setting substantially determines the quality of the traversing movement of an axis. To achieve a sufficient control reserve of analog drives, 80 ... 90% of the desired maximum velocity of the axis should be reached at MD: RATED_OUTVAL.

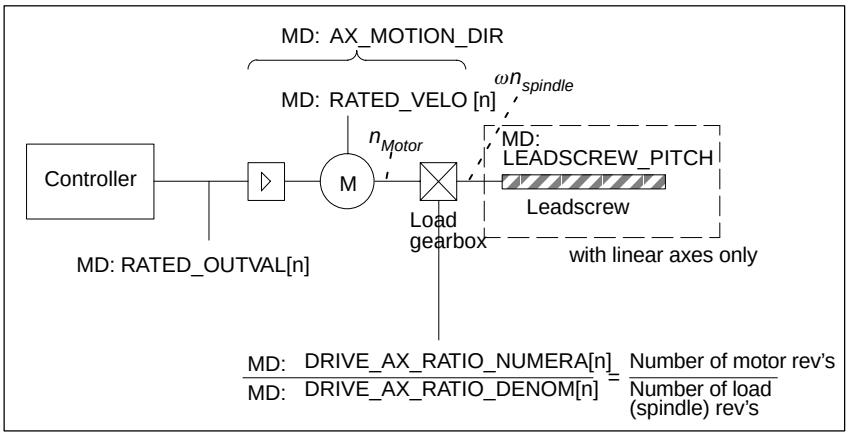


Fig. 3-2 Speed setpoint processing

Example of velocity matching with linear axis

An axis velocity of 15,000 mm / min is to be achieved.
Leadscrew pitch: s = 10 mm/U
Gearbox (motor rev's /spindle rev's) R = 2 : 1 = 2

$$\Rightarrow n_{Motor} = \frac{V_{Achs} * R}{S} = \frac{15000\,\frac{mm}{\min} * 2}{10\,\frac{mm}{rev}} = 3000\ rev/min\ ;$$

The calculated motor speed must be entered in MD: RATED_VELO[n].

⇒ Machine data settings:

MD: RATED_VELO[0] = 3,000 [rev./min]
MD: RATED_OUTVAL = 80 [%] (with analog drives only)
MD: DRIVE_AX_RATIO_NUMERA[0] = 2
MD: DRIVE_AX_RATIO_DENOM[0] = 1
MD: LEADSCREW_PITCH = 10 [mm/rev.]

Actual–value processing

Actual–value resolution

To ensure a correctly closed position control loop, the control system must be informed of the actual–value resolution.

The following axis–specific machine data serve this purpose. The MD marked with * do not apply to stepper motor axes not equipped with an encoder.

The control system automatically calculates the actual–value resolution from the settings made in the machine data.

MD: ENC_IS_DIRECT[n] *	(encoder directly installed on the machine)
MD: DRIVE_ENC_RATIO_DENOM[n]*	(denominator of load gearbox)
MD: DRIVE_ENC_RATIO_NUMERA[n]*	(numerator of load gearbox)
MD: DRIVE_AX_RATIO_DENOM[n]	(denominator of load gearbox)
MD: DRIVE_AX_RATIO_NUMERA[n]	(numerator of load gearbox)
MD: STEP_RESOL	(steps per stepper motor revolution)
MD: ENC_RESOL[n]	(increments per revolution, with stepper motor axes = MD STEP_RESOL)
MD: LEADSCREW_PITCH	(leadscrew pitch)
MD: MAX_AX_VELO	(maximum axis velocity)

The machine data index [n] is coded as follows:
– MD: DRIVE_AX_...[control parameter block no.] : 0–5
– remaining MDs [encoder no.] : 0

Note

These MDs are not required for encoder matching (path evaluation). However, they must be entered correctly for setpoint calculation! Otherwise, the desired servo gain factor (K_V) will not be set.
In MD: DRIVE_AX_RATIO_DENOM the load revolutions are entered, and in MD: DRIVE_AX_RATIO_NUMERA the motor revolutions.

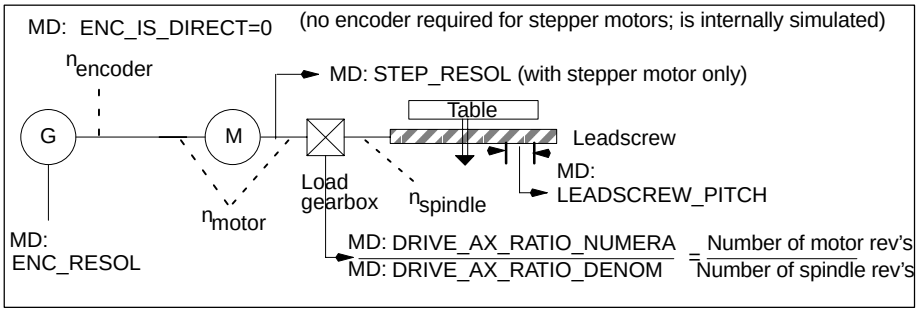


Fig. 3-3 Example: Linear axis with rotary encoder mounted on the motor

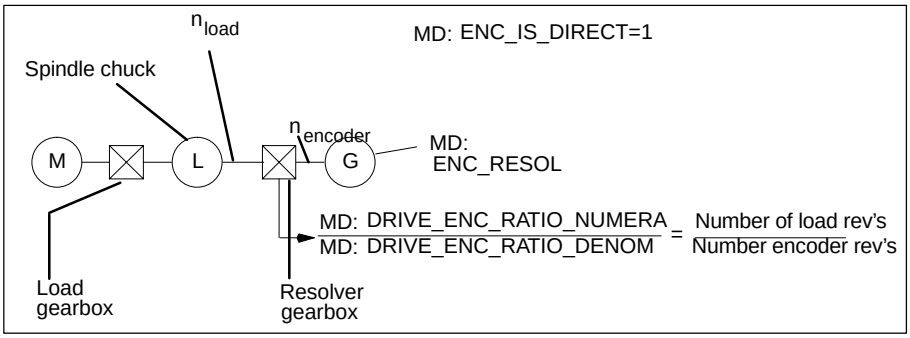


Fig. 3-4 Example: Spindle with rotary encoder mounted on the machine

Note:
MD: ENC_FEEDBACK_POL (actual value sign) can be used to change the sign of actual value sensing and thus the control direction of the position control.

3.2 Closed–Loop Control/Servo Gain

Servo gain factor For stepper motor axes, the entered standard value set in MD: POSCTRL_GAIN[n] should be kept.
For analog axes/spindles, the value should be adapted accordingly.

The machine data index [n] is coded as follows:
[control parameter record no.]: 0 ... 5

However, if the servo gain factor is too high, instability, overshooting and possible impermissibly high loads on the machine will result.

The maximum permissible servo gain factor depends on the following:

- Design and dynamics of the drive
  (rise time, acceleration and deceleration capabilities)
- Quality of the machine (elasticity, vibration suppression)
- Position control cycle

The servo gain factor is defined as follows:

$$KV = \frac{Velocity}{Following\ error}\ ;\ \frac{[m/min]}{[mm]}$$ *Unit of servo gain factor to VDI standard*

Parameter records of the position controller

The position control can use 6 different parameter records. They serve for quick adaptation of the position control to modified properties of the machine during operation, e.g.
– in the case of spindle gear change;
– adaptation of the dynamic properties of an axis, e.g. on tapping.

The following machine data can be changed by switching over the parameter record during operation.
MD: DRIVE_AX_RATIO_DENOM[n] (load gearbox denominator)
MD: DRIVE_AX_RATIO_NUMERA[n] (load gearbox numerator)
MD: POSCTRL_GAIN[n] (servo gain factor)
MD: AX_VELO_LIMIT[n] (velocity monitoring threshold value)
MD: DYN_MATCH_TIME[n] (time constant of dynamic accommodation)
The machine data index [n] is coded as follows:
[control parameter record]: 0–5

Parameter records for the spindle: With the spindle, each gear stage is assigned its own parameter record. Depending on the IS ”Actual gear stage” (V380x2000.0 bis .2), the corresponding parameter record is activated.

Parameter records for axes: For axes not involved in tapping or thread cutting, parameter record 1 is activated (index=0) in all cases. For axes involved in tapping or thread cutting, the same parameter record number is activated as with the current gear stage of the spindle.

3.3 Velocity Control for Stepper Motors

3.3.1 Knee–Shaped Acceleration Characteristic

Stepper motor drives only with SINUMERIK 802S!

A characteristitc feature of stepper drives is the decay of the available torque in the upper speed range (see Fig. 3-5).

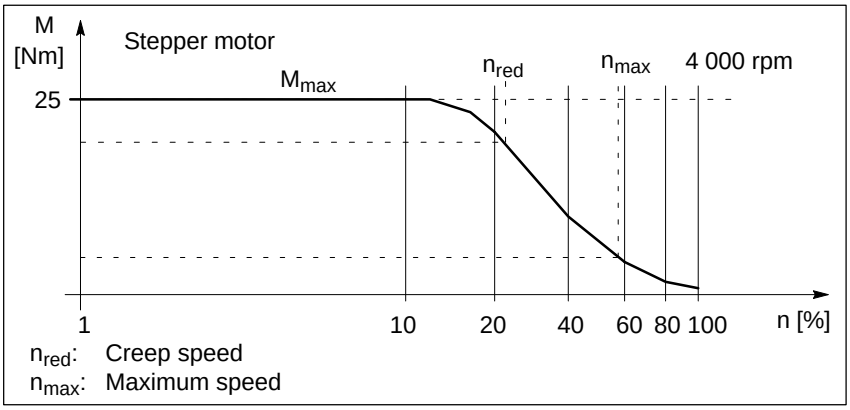


Fig. 3-5 Typical motor characteristics of a stepper drive

The optimum utilization of such characteristics with overload protection at the same time can be achieved with velocity–dependent acceleration control.

This method called ”knee–shaped acceleration characteristic” can be used for both positioning and path movements.

Parameterization of the axis characteristic

The axis–specific course of the acceleration characteristic must be parameterized using the following machine data:

- **MD: MAX_AX_VELO** (
  Maximum axis–specific velocity (v_{max})
- **MD: ACCEL_REDUCTION_SPEED_POINT**
  Threshold velocity of the acceleration decay with respect to: MAX_AX_VELO (v_{red})
- **MD: MAX_AX_ACCEL**
  Maximum axis–specific acceleration (a_{max})
- **MD: ACCEL_REDUCTION_FACTOR**
  Factor of acceleration reduction with respect to
  MD: MAX_AX_ACCEL (a_{red})
- The acceleration course is constant.

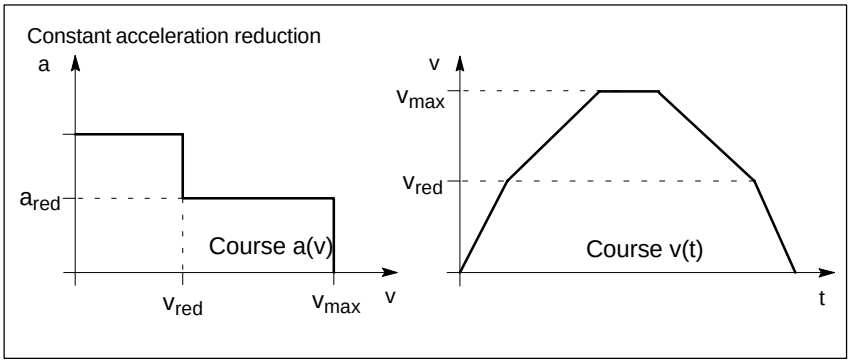


Fig. 3-6 Axis–specific acceleration and velocity course

Velocities:
v_{max}: MD: MAX_AX_VELO
v_{red}: MD: ACCEL_REDUCTION_SPEED_POINT x MD: MAX_AX_VELO

Accelerations:
a_{max}: MD: MAX_AX_ACCEL
a_{red}: (1–MD: ACCEL_REDUCTION_FACTOR) x MD: MAX_AX_ACCEL

Activation

Traversing the stepper motor axes in JOG mode:
This feature is always activated with MD: ACCEL_TYPE_DRIVE = 1.
MD: JOG_AND_POS_JERK_ENABLE=0 must be set.

The knee–shaped acceleration characteristic is an axis–specific default setting for all stepper motor axes (SINUMERIK 802S).

Path movement (G1, G2, G3, ...):
With the SINUMERIK 802S, the activation of the knee–shaped acceleration characteristic for the path movement is carried out automatically when turning on by setting an internal machine data to the turn–on setting of the G command DRIVE. A switchover/deselection is not possible in the program.
MD: ACCEL_TYPE_DRIVE has no influence here.

Path characteristic

No additional machine data exist for the path movement.
The characteristic comprises of the parameters of the axes involved, depending on their portion in the path vector (geometry).
A combination of axes with different acceleration courses is permitted.

Normal and tangent accelerations within knee–shaped path sections are discussed together.

The path velocity is reduced as far as a maximum of 25 % of the velocity–dependent acceleration capability of the axes is required for the normal acceleration. The residue is reserved for tangential acceleration, i.e. braking and/or acceleration on the path.

G64 block transition

Axis–specific velocity steps can occur at non–tangential block transitions.

The path velocity at the block transition is reduced if an axis–specific velocity portion is above the threshold velocity of the acceleration decay
(MD: ACCEL_REDUCTION_SPEED_POINT).

3.3.2 Parameterization of the Stepper Motor Frequency

Stepper motor frequency

The maximum stepper motor frequency is defined with the machine data

MD: FREQ_STEP_LIMIT [Hz]:

$$\text{MD value} = \frac{\text{Motor speed [rpm]} \cdot \text{steps per } 360°}{60\ \text{[s]}}$$

This frequency must correspond to the MD: MAX_AX_VELO (axis velocity).

Example: MD: Determining MAX_AX_VELO and MD: FREQ_STEP_LIMIT

$$\frac{\text{Motor speed [rpm]} \cdot \text{Leadscrew pitch [mm/rev]}}{\text{Load gearbox}} = \text{Axis velocity}$$

Motor speed:	1 200 rpm
Load gearbox (R):	1:1 –>R=1
Leadscrew pitch:	10 mm
Steps per 360°:	10 000

$$\frac{1\ 200\ \text{rpm} \cdot 10\ \text{mm}}{1} = 12\ 000\ \text{mm/min} \text{ –> MD: MAX_AX_VELO}$$

The resulting frequency limit must be:

$$\frac{1\ 200\ \text{rpm} \cdot 10\ 000\ \ 1/\text{rev}}{60\ \text{s}} = 200\ 000\ \text{Hz} \text{ –> MD: FREQ_STEP_LIMIT}$$

Stepper motor without encoder

When a stepper motor **without** encoder is used,

the number of steps per 360° must also be entered in MD: ENC_RESOL.

Example:

Stepper motor: 10 000 [pulses per motor revolution]
Load gearbox: 1:1
Leadscrew pitch: 10 mm
Motor speed: 1 200 rpm

The following MD values result from this:

MD: CTRLOUT_TYPE = 2 (setpoint output for stepper motor)
MD: ENC_TYPE = 3 (stepper motor without encoder)
MD: ENC_RESOL[0] = 10 000 (no pulse quadrupling)
MD: STEP_RESOL = 10 000
MD: FREQ_STEP_LIMIT[Hz] = 200 000 [Hz]
MD: MAX_AX_VELO = 12 000 mm/min

Stepper motor with encoder

If the stepper motor is operated with encoder, the encoder adaptation has to be carried out as with analog drives. To determine the adaptation, it should be taken into account that the encoder pulses are quadrupled.

3.4 Data Description

10240 MD number	SCALING_SYSTEM_IS_METRIC Metric scaling system		
Default: 1	Min. input limit: 0	Max. input limit: 1	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	This MD defines the system for scaling length–dependent physical quantities used by the control system for data input/output. Internally, all data are stored in the units 1 mm, 1 degree and 1 sec. When accessing from the part program, from the operator panel or via external communication, the scaling is carried out with the following units: SCALING_SYSTEM_IS_METRIC = 1: scaled in: mm, mm/min, m/s^2, m/s^3, mm/rev. SCALING_SYSTEM_IS_METRIC = 0: scaled in: inch, inch/min, inch/s^2, inch/s^3, inch/rev. The choice of the scaling system also defines the interpretation of the programmed F value for linear axes: metric / inch G94 mm/min / inch/min G95 mm/rev. / inch/rev. After this machine data has been changed, the control system must be rebooted; otherwise, related machine data that have physical units will be scaled not correctly. Observe the following procedure: • MD change by manual input: ⇒ Reboot and then enter the appropriate machine data with their physical units. • MD are changed via the machine data file ⇒ Reboot and then reload machine data file to make sure that the new physical units are taken into account. When the machine data is changed, alarm 4070 ”Scaling machine data changed” is output.		

30130 MD number	CTRLOUT_TYPE[n] Output type of setpoint		
Default: 0	Min. input limit: 0	Max. input limit: 4	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BYTE	Valid as from SW version:		
Meaning:	This MD is used to enter the type of the speed setpoint output: **0:** Simulation (no HW required) **1:** Standard (differentiation via HW configuration) **2:** Stepper motor 3, 4: not available The machine data index [n] is coded as follows: [setpoint branch]: 0		
Application example(s)	Simulation: Machine functions can also be simulated when the drive is not connected.		

30200 MD number	NUM_ENCS Number of encoders		
Default: 1	Min. input limit: 0	Max. input limit: 1	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BYTE	Valid as from SW version:		
Meaning:	This MD is only necessary if the position actual value sensing is to be carried out using a direct measuring system (i.e. not with a motor–installed measuring system and not with stepper motors). **1:** Spindle/axis with direct measuring system (on the machine) **0:** Spindle without measuring system		

30240 MD number	ENC_TYPE[n] Mode of actual–value sensing (actual position value)		
Default: 0	Min. input limit: 0	Max. input limit: 5	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BYTE	Valid as from SW version:		
Meaning:	In this MD, the encoder type used must be entered: **0:** Simulation **2:** Square–wave generator (standard, quadrupling of increments) **3:** Encoder for stepper motor (Values: 1, 2, 5: not available) The machine data index [n] is coded as follows: [encoder no.]: 0 If an invalid encoder type is defined, alarm 300009, ”Invalid measuring circuit type – drive [number], measuring circuit [number]” is output.		
Application example(s)	Simulation: Machine functions can also be simulated with a measuring system connected.		

30350 MD number	SIMU_AX_VDI_OUTPUT Output of axis signals with simulation axes		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	This machine data defines whether axis–specific interface signals are output to the PLC during simulation. **1:** The axis–specific IS of a simulated axis are output to the PLC. It is thuis possible to test the PLC user program without drives connected. **0:** The axis–specific IS of a simulated axis are not output to the PLC. All axis–specific IS are set to “0”.		
MD irrelevant bei	MD: CTRLOUT_TYPE (output mode of setpoint) = 1		
Application example(s)	MD: SIMU_AX_VDI_OUTPUT = 0 For example, this prevents that the brake is opened when simulating an axis.		

31000 MD number	ENC_IS_LINEAR[n] Direct measuring system (linear scale)		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Change effective after Power On	Protection level: 2	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	**1:** The encoder for actual position sensing is linear (linear scale). **0:** The encoder for actual position sensing is rotary. The machine data index is coded as follows: [encoder no.]: 0		
Further references			

31020 MD number	ENC_RESOL[n] Increments per revolution		
Default: **802S:** (1000, 1000, 1000, 2048) **802C:** (2500, 2500, 2500, 2048)	Min. input limit: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: DWORD	Valid as from SW version:		
Meaning:	In this MD, the increments per encoder revolution must be entered. The machine data index is coded as follows: [encoder no.]: 0		

31030 MD number	LEADSCREW_PITCH Leadscrew pitch		
Default: 10	Min. input limit: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: mm/Umdr.	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In this MD, the leadscrew pitch is entered.		

31040 MD number	ENC_IS_DIRECT[n] Encoder is directly mounted on the machine		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	**1:** The encoder for actual value sensing is directly mounted on the machine. **0:** The encoder for actual value sensing is mounted on the motor. The machine data index is coded as follows: [encoder no.]: 0		
Special cases, errors, ...	Invalid or illegal values can result in erroneous encoder resolutions, since e.g. incorrect gear ratios are taken into account.		

31050 MD number	DRIVE_AX_RATIO_DENOM[n] Load gearbox denominator	
Default: 1	Min. input limit: 1	Max. input limit: 2 147 000 000
Change effective after Power On	Protection level: 2/7	Unit: –
Data type: DWORD	Valid as from SW version:	
Meaning:	In this MD, the denominator of the load gearbox must be entered. The machine data index is coded as follows: [control parameter record]: 0–5	
Further references		

31060 MD number	DRIVE_AX_RATIO_NUMERA[n] Load gearbox denominator	
Default: 1	Min. input limit: 1	Max. input limit: 2 147 000 000
Change effective after Power On	Protection level: 2/7	Unit: –
Data type: DWORD	Valid as from SW version:	
Meaning:	In this MD, the numerator of the load gearbox must be entered. The machine data index is coded as follows: [control parameter record]: 0–5	

31070 MD number	DRIVE_ENC_RATIO_DENOM[n] Resolver gearbox denominator	
Default: 1	Min. input limit: 1	Max. input limit: 2147000000
Change effective after Power On	Protection level: 2/7	Unit: –
Data type: DWORD	Valid as from SW version:	
Meaning:	In this MD, the denominator of the resolver gearbox must be entered. The machine data index is coded as follows: [encoder no.]: 0	

31080 MD number	DRIVE_ENC_RATIO_NUMERA[n] Resolver gearbox numerator	
Default: 1	Min. input limit: 1	Max. input limit: 2147000000
Change effective after Power On	Protection level: 2/7	Unit: –
Data type: DWORD	Valid as from SW version:	
Meaning:	In this MD, the numerator of the resolver gearbox must be entered. The machine data index is coded as follows: [encoder no.]: 0	

31400 MD number	STEP_RESOL Steps per stepper motor revolution		
Default: 1000	Min. input limit: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: DWORD	Valid as from SW version:		
Meaning:	Output parameterization for stepper motor		

32000 MD number	MAX_AX_VELO Maximum axis velocity		
Default: 10000	Min. input limit: 0	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: mm/min, Umdr./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In this MD, the limit velocity up to which the axis can accelerate must be entered (rapid traverse limiting). If rapid traverse is programmed, this velocity is used for traversing. The maximum admissible axis velocity is dependent on the dynamic properties of machine and drive, as well as on the limit frequency of the actual value sensing.		

32100 MD number	AX_MOTION_DIR Traversing direction		
Default: 1	Min. input limit: –1	Max. input limit: 1	
Change effective after Power On	Protection level: 2/2	Unit: –	
Data type: BYTE	Valid as from SW version:		
Meaning:	This MD can be used to reverse the traversing direction of the machine. The control direction, however, is not reversed during this process, i.e. the control remains stable. **0 or 1:** no direction reversal **–1:** Direction reversal		

32110 MD number	ENC_FEEDBACK_POL[n] Actual value sign (control direction)		
Default: 1	Min. input limit: – 1	Max. input limit: 1	
Change effective after Power On	Protection level: 2/2	Unit: –	
Data type:BYTE	Valid as from SW version:		
Meaning:	In this MD, the evaluation direction of the encoder signals is entered. **0 or 1**: No direction reversal **–1:** Direction reversal The direction reversal also pertains to the control direction if the encoder is used for the position control. The machine data index is coded as follows: [encoder no.]: 0		
Special cases, errors, ...	If the wrong control direction is entered, the axis can run away. Depending on the setting of the corresponding limit values, one of the following alarms is output: Alarm 25040 ”Zero speed control” Alarm 25050 ”Contour monitoring” Alarm 25060 ”Speed setpoint limiting“ The corresponding limit values are described in: **References:** Chapter ”Axis Monitoring Functions“ If an uncontrolled setpoint step occurs when a drive is connected, the control direction is possibly wrong.		

32200 MD number	POSCTRL_GAIN[n] Servo gain factor		
Default: **802S:** (2.5, 2.5, 2.5, 1) **802C:** (1, 1, 1, 1)	Min. input limit: 0	Max. input limit: 2000	
Change effective after NEW_CONF	Protection level: 2/7	Unit: 1/s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	(Do not change with stepper motor axes!) Position control gain, so–called KV factor (servo gain factor) The input/output unit for the user is [(m/min)/mm]. This means that POSCTRL_GAIN[n] = 1 corresponds to a following error of 1 mm with V = 1 m/min. If “0” is entered, the position controller is disconnected. When entering the servo gain factor (”KV factor”), take into account that the gain factor of the whole position control loop also depends on other parameters of the controlled system. It must therefore be distinguished between a “desired” servo gain factor (MD: POSCTRL_GAIN) and a ”real servo gain factor” (that results on the machine). Only if all parameters of the control loop are matched one to another correctly, these servo gain factors are identical. **Note:** Interpolating axes that are to carry out a machining must possess the same gain (i.e. the same following error at the same velocity). The real KV factor (servo gain factor) can be checked using the following error (in the service displays). The machine data index is coded as follows: [control parameter record no.]: 0–5		

32250 MD number	RATED_OUTVAL[n] Rated output voltage		
Default:(80, 80, 80, 100)	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: %	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	(Not with stepper motor axes!) In this MD, enter the value of the speed setpoint at which the motor speed specified in MD: RATED_VELO[n] is reached as a percentage.		
Application example(s)	**1st example:** At a voltage of 5 V, the drive reaches a speed of 1,875 rpm. ⇒ RATED_OUTVAL = 50 %, RATED_VELO = 1,875 [rpm] **2nd example:** At a voltage of 8 V, the drive reaches a speed of 3,000 rpm. ⇒ RATED_OUTVAL = 80 %, RATED_VELO = 3,000 [rpm] **3rd example:** At a voltage of 1.5 V, the drive reaches a speed of 562.5 rpm. ⇒ RATED_OUTVAL = 15 %, RATED_VELO = 562.5 [rpm] All three examples above are possible for one and the same drive/inverter. The ratio of the two values another to one is decisive, and this is the same in all three examples. The machine data index [n] is coded as follows: [setpoint branch]: 0		
Related to	MD: RATED_OUTVAL[n] only makes sense in conjunction with MD: RATED_VELO[n].		

32260 MD number	RATED_VELO[n] Rated motor speed		
Default: 3000	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: U/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In this MD, enter the speeed of the drive (standardized on the part of the drive!) which is reached at the percentage speed specified in MD: RATED_OUTVAL[n]. The machine data index [n] is coded as follows: [setpoint branch]: 0		
Related to	MD: RATED_VELO[n] only makes sense in conjucntion with MD:RATED_OUTVAL[n].		

32900 MD number	DYN_MATCH_ENABLE Dynamic response adaptation		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Change effective after NEW_CONF	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	The dynamic response adaptation can be used to set axes having different servo gain factors to the same following error using MD: DYN_MATCH_TIME. **1:** Dynamic response adaptation is enabled. **0:** Dynamic response adaptation is disabled.		
Application example(s)			
Related to	MD: DYN_MATCH_TIME[n] (time constant of dynamic response acquisition)		

32910 MD number	DYN_MATCH_TIME[n] Time constant of dynamic response adaptation		
Default: 0.01	Min. input limit: 0	Max. input limit: plus	
Change effective after NEW_CONF	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In this MD, the time constant of the dynamic response adaptation of an axis must be entered. Enter the difference of the equivalent time constant of the 'slowest' control loop of the corresponding axis as the time constant. The MD is only effective if MD: DYN_MATCH_ENABLE = 1. The machine data index is coded as follows: [control parameter record no.]: 0–5		
Application example(s)	see Section 2.3		
Related to	MD: DYN_MATCH_ENABLE (dynamic response adaptation)		

Manual Traversing and Handwheel Traversing 4

Setting up the machine

Even modern numerically controlled machine tools must allow the axes to be traversed manually. In particular, when setting up a new machining program, it is necessary to move the axes either by means of the traversing keys on the machine control panel or the electronic handwheel.

Clearing the tool

After interrupting the program by certain events, such as NC–STOP, RESET or mains power failure, the machine operator must clear the tool from the current machining position manually. This is usually done by means of the direction keys in JOG mode.

Contents

The present Functional Description describes the following possibilities and features of manual traversing:

- Continuous traversing in JOG mode
- Incremental method (INC) in JOG mode
- Traversing the axes using electronic handwheels (accessories) in JOG mode

4.1 General Properties of Manual Traversing in JOG Mode

The following paragraphs will describe the generally applicable properties of manual traversing in JOG mode (irrespective of the selected variant):

JOG mode

To traverse the axes manually (further called 'manual traversing'), JOG mode must be active. The currently active operating mode is reported to the PLC via the interface signal (IS) "Active mode: JOG" (V30000000.2).

Machine functions

JOG mode divides into several JOG variants (machine functions):

- continuous traversing
- incremental traversing
- traversing with handwheel

Traversing

Axes can be traversed in the following coordinate systems:

- Machine coordinate system (MCS)
- Workpiece coordinate system (WCS)

The currently active machine function is selected via the PLC interface. There are separate PLC interfaces both for the axes in MCS (axis–specific) and for the axes in WCS (channel–specific).

Simultaneous traversing

In JOG mode, all axes can be traversed simultaneously.
When the axes are traversed simultaneously, the axes do not interpolate with each other.

Velocity

The velocity of the traversing movement in JOG mode is determined by the following values specifications:

SD: JOG_SET_VELO (JOG velocty with G94) for one axis,
SD: JOG_SPIND_SET_VELO (JOG velocity for spindle)
If the value of this SD is zero, the value of MD: JOG_VELO (conventional axis velocity) is used. To limit the axis velocity, MD: MAX_AX_VELO is used.

Rapid traverse override

If the rapid traverse override key is pressed in addition to the traversing keys, the movement is carried out with the rapid traverse speed set via the axis–specific MD: JOG_VELO_RAPID (axis speed in JOG mode with rapid traverse override).

Feed override

The axis velocity used for traversing in JOG mode can additionally be controlled using the axis–specific feed override switch provided the axis–specific IS “Override enabled” (V380x0001.7) is set.

Acceleration/jerk

The axis acceleration is defined with the axis–specific MD: MAX_AX_ACCEL. With manual traversing, too, the acceleration is possible according to a given characteristic. The acceleration curve for the individual axes in JOG mode is defined with MD:
JOG_AND_POS_MAX_JERK (acceleration change limited) provided it is activated with MD:
JOG_AND_POS_JERK_ENABLE = 1.

For stepper motor axes, it is recommended to use the knee–shaped acceleration curve. This is the default setting for SINUMERIK 802S.
It is enabled with MD: ACCEL_TYPE_DRIVE = 1. When doing so,
MD: JOG_AND_POS_JERK_ENABLE=0 must be set. Other machine data are provided to set the characteristic curve (see also Section 3.3.1).

4.2 Controlling Manual Traversing via the PLC Interface

MMC/NCK/PLC interface

The individual functions for manual traversing in JOG mode are mainly activated via the PLC interface.

MMC/NCK/PLC interface

In particular, the following signals of the machine control panel (MCP) are relevant for manual traversing:

- JOG mode (selection)
- Machine functions INC1 , ...
- Direction keys
- Feed override or spindle override

Example: Spindle speed in JOG mode

If a spindle speed is to be specified for an analog spindle in JOG mode by operating a key on the machine control panel to rotate it in the specified direction or to stop it, use the following procedure:

Choose the keys for “Spindle CCW”, “Spindle CW” and “Spindle Stop” from the keys beneath the free keys on the machine control panel. The PLC user program must assign the incoming key signals to the IS “Traversing key plus” or “Traversing key minus” (V38030004.7 or .6), observing the following prerequisites:

- Only one of the signals may be set.
- The signal remains also set when the key is released.
- If the Spindle Stop key is pressed, both traversing signals must be cleared.
- Changing from “Traversing key plus” to “Traversing key minus” or vice versa is only possible via the Spindle Stop status (both traversing signals cleared).
- If the IS “Reset” (V30000000.7) is provided, the spindle is stopped, and the traversing signals must be cleared.

The spindle speed is set via operation using a setting data item. Access to the setting data defining the spindle speed JOG SD: JOG_SPIND_SET_VELO is granted via the Setting Data menu. If this value =0, the value in MD: JOG_VELO will also apply for the spindle.

When traversing the spindle in JOG mode, the maximum speeds of the active gear stage (MD: GEAR_STEP_VELO_LIMIT) are taken into account.

4.3 Continuous Traversing

Selection

When JOG is selected, the machine function: “continuously active” is automatically set in the axes in WCS and in the machine axes
(IS: V33001001.6, V33001005.6, V33001009.6, V390x0005.6).
In JOG mode, it is also possible to activate continuous traversing via the PLC interface (IS ”Machine function: continuous“ for the axes in WCS (V32001001.6, VB32001005.6, VB32001009.6) and for the machine axes (VB380x0005.6).

Deselection

The Continuous Traversing function is deselected by selecting incremental traversing (see Section 4.4).

Traversing keys +/–

The Plus and Minus traversing will traverse the related axes in the desired directions (PLC at NCK interface: V32001000.7/.6, V32001004.7/.6, V32001008.7/.6 or V380x004.7/.6).
If both traversing keys of an axis are actuated at the same time, no traversing movement is carried out or the axis currently moving is stopped.

The axis will traverse as long as the traversing key signal is set if no axis limitation is reached beforehand. When the signal is reset, the axis is decelerated to a standstill, and the movement is considered ended.

Traversing commands +/–

Once a traversing request is present for an axis, the IS “Traversing command +” or “Traversing command –” (V33001000.7/.6, V33001004.7/.6, V33001008.7/.6 or V390x004.7/.6)) is output to the PLC, depending on the direction of the movement.

4.4 Incremental Traversing (INC)

Setting increments

The path to be traversed by the axis is determined by so–called increments (also called “incremental dimension”). Before the machine operator can traverse the axis, he must set the desired increment.
The setting is done via the machine control panel.

Settable increments

The operator can set up max. four different increment steps that apply to all axes together: INC1, INC10, INC100 and INC1000.

Increment weighting

The axis MD: JOG_INCR_WEIGHT (weighting of an axis increment with INC/handwheel) is used to define the path weighting of **one** JOG increment. Default setting is 1 incr.= 0.001 mm.

Traversing

When the traversing key of the desired direction (e.g. +) is pressed, the axis starts traversing with the set increment. When the traversing key is released before the increments have been traversed completely, the movement is interrupted and the axis stops. When the same traversing key is pressed once more, the axis traverses the remaining distance to go until it is zero. The movement can be interrupted by releasing the traversing key.
Pressing the traversing key of the opposite direction remains without effect as long as the increment has not been traversed completely or the movement is aborted.

Canceling the traversing movement

If you do not wish to traverse the entire increment, the movement can be canceled either by pressing RESET or providing the axis interface signal ”Clear distance to go/Spindle Reset” (V380x0002.2).

4.5 Handwheel Traversing in JOG Mode

Selection

JOG mode must be active. In addition, the operator must set the increment active during handwheel traversing INC1, INC10, ... The assignment axis/handwheel must be carried out on the operator panel (see Documentation “Operation and Programming“).

Traversing

Turning the electronic handwheel traverses the respective axis in positive or negative direction (depending on the desired direction of rotation).

Travel or velocity specifications

Default settings for the handwheel movement to match it with the intended application can be defined using MD: HANDWH_TRUE_DISTANCE (handwheel travel or velocity specification):

Value=1 (standard): The handwheel default values are travel specifications. No pulses will be lost. Due to a limitation to the maximum admissible velocity, it is possible that the axes follow up. This should be taken into account, in particular, in the case of a high weighting of the handwheel pulses.

Value=0: The handwheel default values are velocitiy specifications. Decelerating in the case of a handwheel standstill will be carried out using the shortest way.

Weighting

The distance to be traversed, which results from turning the handwheel, depends on the following factors:

- number of handwheel pulses received on the interface
- active increment (machine functions INC1, INC10, INC100, ... INC1000)
- pulse weighting of handwheel with general MD: HANDWH_IMP_PER_LATCH (handwheel pulses per latched position)
- weighting of an increment with INC/handwheel (axis–specific MD: JOG_INCR_WEIGHT).

Traversing commands +/-

During the axis movement, either the IS “Traverse command+” or “Traverse command–” (V380x0004.7 or .6) is provided to the PLC, depending on the direction of movement.
If the axis is already traversed via the traversing keys, additional handwheel traversing is not possible. Alarm 20051 ”Handwheel traversing not possible” is output.

Connection of handwheels

A maximum of 2 handwheels can be connected at a time. It is thus possible to move max. 2 axes by handwheels at the same time.

Handwheel assignment

It is possible to assign an axis a handwheel either in the machine (MCS) or in the workpiece (WCS) coordinate system.

Which axis (X, Y, Z) is moved by turning handwheel 1 or 2 can be set via menu–assisted operation (MMC):

When the **handwheel** softkey is pressed in the basic menu of JOG mode, the Handwheel window is displayed. In this window, each handwheel can be assigned an axis and the handwheel be enabled or disabled.

Handwheel selection from MMC

To activate the handwheel from the operator panel, special data in the user interface between MMC and PLC are used. This interface made available from the PLC basic program for handwheels 1 and 2 can be monitored and contains the following information (the respective axis name X, Y, Z is replaced by an axis number (1,3):

- the axis number assigned to the handwheel
  IS ”Axis number of handwheel 1” (VB19001003) IS ”Axis number of handwheel 2” (VB19001004)
- the additional information “Machine axis” or “Axis in WCS”
  IS (V19001003.7 or V19001004.7)

Input frequency

The handwheel connections can receive handwheel pulses with a maximum input frequency of 100 kHz.

Velocity

The velocity results from the pulses generated using the handwheel, and from the pulse weighting: Distance to be traversed per time unit
This velocity is limited by the value set in the axis–specific
MD: MAX_AX_VELO.

Acceleration

When traversing using the handwheel, the acceleraiton (axis with analog drive or stepper motor axis) is depending on the acceleration characteristic defined for JOG in the axis–specific machine data (see Section LEERER MERKER).

Canceling the traversing movement

RESET or axis IS ”Delete distance to go/Spindle Reset” (V380x0002.2) will cancel the traversing movement. The existing set/actual difference is deleted. NC STOP will only interrupt the traversing movement. Any set/actual difference remains. The distance to go is then covered by NC START.

Traversing in the opposite direction

Depending on the machine data: HANDWH_REVERSE, the response in case of traversing direction reversal is as follows:

- If the handwheel is moved in the opposite direction, the resulting distance to be traversed is calculated and the end point calculated in this way approached as fast as possible: If this end point is ahead of the point to which the moving axis can decelerate with the current traversing direction, deceleration is carried out and the end point approached in the opposite direction. Otherwise, the newly calculated end point is approached immediately.
- If the handwheel is moved in the opposite direction by at least the number of pulses specified in the machine data, the axis is decelerated as fast as possible, and all pulses that come until the end of interpolation are ignored, i.e. the axis is traversed again only after it has come to standstill (on the setpoint end).

Response at the software limit switch

When traversing in JOG mode, the axes are only traversed to the respectively first active limit switch; then the respective alarm is output. Depending on the machine data HANDWH_REVERSE, the response is as follows (as long as the axis has not reached the end point as far as the setpoint is concerned):

- The traverse path resulting from the handwheel pulses forms an effective end point used for the calculations to follow: If this fictive end point is, for example, is 10 mm behind this limit, these 10 mm must first be traversed in the opposite direction before the axis can carry out a real movement. If you wish to traverse at a limit immediately in the opposite direction again, the fictive distance to go can be cleared either by “Clear distance to go” or deselecting the handwheel assignment.
- All handwheel pulses that lead to an end point behind the limit are ignored. Moving the handwheel immediately in the opposite direction results in traversing in the opposite direction, i.e. from the limit switch away.

4.6 Special Features of Handwheel Traversing

4.6.1 Monitoring Functions

Limit switches

The following limit switches are used for handwheel traversing:

- Software limit switches 1 and 2 (axis must be referenced)
- Hardware limit switches

The control system has been conceived such that the traversing movement is aborted as soon as the first valid limit switch is reached. The velocity control ensures that the deceleration process is started on time so that the axis stops exactly on the limit position (e.g. software limit switches). Fast Stop is only used to stop the axis if the hardware limit switches respond.

If the respective limit switch is reached, an alarm message (alarms 10620, 10621) is output. The control system then prevents further traversing in this direction. The traversing keys and the handwheel for this direction remain without effect.

Important

In order to activate the software limit switches, the axis must first be referenced.

Clearing the axis

The axis can be traversed from a limit switch position to the opposite direction.

Machine manufacturer

The way how an axis that has approached the limit position is cleared depends on the machine manufacturer. Please refer to the Documentation of the machine manufacturer!

Maximum velocity and acceleration

The velocity and acceleration used for manual traversing is defined axis–specifically by the start–up engineer via machine data. The control system limits the values acting at the axes to the maximum velocity and acceleration settings.

4.6.2 Miscellaneous

Mode change JOG→AUT or JOG→MDA

Mode change from JOG to AUT or MDA is only carried out if all axes have reached Exact Stop Coarse.

Transverse axes

The X axis is the transversal axis. When traversing this axis in JOG mode, the following should be considered:

- Continuous traversing:
  When a transversal axis is traversed continuously, there are no differences.
- Incremental traversing:
  Only the **half of the distance** of the selected increment size is traversed. For example, when the traversing key is pressed and INC10 is set, the axis traverses by 5 increment values with refer to the radius (10 diameter increments).
- Traversing with the handwheel:
  Only the half of the distance per handwheel pulse is traversed when the axes are traversed by increments using the handwheel.

4.7 Data Description

Machine data

11310 MD number	$MN_HANDWH_REVERSE Threshold for change of handwheel direction		
Default: 2	Min. input limit: 0	Max. input limit: –	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BYTE	Valid as from SW version:		
Meaning:	0: No immediate traversing in opposite direction >0: Immediate traversing in opposite direction if the handwheel is turned in the opposite direction at least by the specified number of pulses		

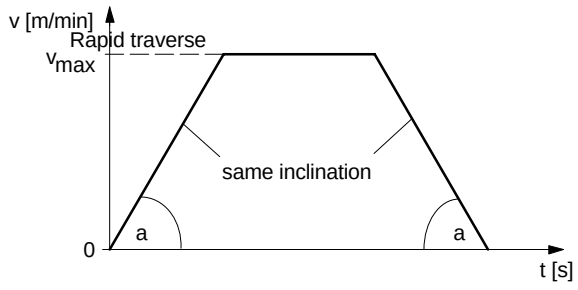
11320 MD number	HANDWH_IMP_PER_LATCH[n] Handwheel pulses per latched position [handwheel number]: 0 ... 1		
Default: 1	Min. input limit: ***	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	MD: HANDW_IMP_PER_LATCH is used to adapt the connected handwheels to the control system. Enter the number of pulses per handwheel latched position generated by the handwheel. The handwheel pulse weighting must be set for each existing handwheel (1 to 3) separately. Using this adaptation, each handwheel latched position acts in the same manner as a traversing key would have been pressed during incremental traversing. A negative value results in direction reversal of the direction of rotation of the handwheel.		
Related to	MD: JOG_INCR_WEIGHT (weighting of an axis increment with INC/handwheel)		

11346 MD number	HANDWH_TRUE_DISTANCE Handwheel travel or velocity specification		
Default: 1	Min. input limit: 0	Max. input limit: 2	
Change effective after Power On	Degree of protection: 2/7	Unit: –	
Data type: BYTE	Valid as from SW version: 3		
Meaning:	**0:** The specifications of the handwheel are velocity specifications. Deceleration at a standstill of the handwheel is on the shortest way. **1:** The specifications of the handwheel are travel specifications. No pulses are lost. Due to a limitation to the maximum admissible velocity, the axes can follow up. **2:** not available		

31090 MD number	JOG_INCR_WEIGHT Weighting of an increment with INC/handwheel		
Default: 0.001	Min. input limit: ***	Max. input limit: ***	
Change effective after Power On	Protection level: 2/7	Unit: Linear axis: mm	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The entered value defines the **traverse distance of an increment** which is used when traversing an axis via the JOG keys either in incremental mode or with the handwheel. The distance traversed by the axis with incremental dimension each time when the key is pressed or per handwheel latched position is defined by the following parameters: • MD: JOG_INCR_WEIGHT (weighting of an axis increment with INC/handwheel) • Selected increment size (INC1, ..., INC1000) A negative value results in a reversal of the direction weighting of the direction keys or of the direction of rotation of the handwheel.		
MD not applicable to	AUTOMATIC mode and MDA		

32010 MD number	JOG_VELO_RAPID Conventional rapid traverse		
Default: 10000	Min. input limit: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: Linear axis: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The entered axis velocity applies to traversing in JOG mode with the Rapid Traverse Override key pressed and axis feed override 100%. The entered value may not exceed the maximum permissible axis velocity (machine data MAX_AX_VELO). These machine data are **not** used for the programmed rapid traverse G00.		
MD not applicable to	AUTOMATIC mode and MDA		
Related to	MD: MAX_AX_VELO (maximum axis velocity) IS ”Rapid traverse override“ IS ”Feed override“		

32020 MD number	JOG_VELO Conventional axis velocity		
Default: 2000	Min. input limit: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: Linear axis: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The entered velocity applies to traversing in JOG mode with axis feed override position on 100%. This velocity is only used if the general setting data for linear axes SD: JOG_SET_VELO = 0. In this case, the axis velocity acts for – continuous traversing – incremental traversing (INC1, ...) – handwheel traversing The entered value may not exceed the maximum permissible axis velocity (machine data MAX_AX_VELO). Spindles in JOG mode: These machine data can also be used to set a spindle–specific velocity for spindles when traversing in JOG mode (provided SD: JOG_SPIND_SET_VELO = 0). However, this velocity is affected by the spindle override switch.		
Application example(s)	If different velocities are required for the axes/spindles in JOG mode, the velocity can be defined axis–specifically. To this aim, SD: JOG_SET_VELO must be set to 0.		
Related to	MD : MAX_AX_VELO (maximum axis velocity) SD: JOG_SET_VELO (JOG velocity for G94) IS ”Feed override“		

32300 MD number	MAX_AX_ACCEL Axis acceleration		
Default: 1.0	Min. input limit: 0.0	Max. input limit: ***	
Change valid after Power On	Protection level: 2/7	Unit: m/s^2, rev/s^2	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The acceleration specifies a velocity change of the axis as a function of the time. Different axes need not have the same acceleration. The lowest acceleration value of the axes involved in the interpolation is taken into account. Ask your machine manufacturer for which continuous braking and continuous acceleration your machine is suited. This value is entered into this machine data. The acceleration value is effective during each acceleration and/or deceleration process.		
MD not applicable to ...	errors resulting in quick stop		
			

32420 MD number	JOG_AND_POS_JERK_ENABLE Default setting of axial jerk limitation		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Change valid after Power On	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version: 3		
Meaning:	enables the function of the axis–specific jerk limiting for the operating modes JOG, REF.		
Related to	MD: JOG_AND_POS_MAX_JERK (axial jerk) MD: ACCEL_TYPE_DRIVE (acceleration reduction ON/OFF)		

32430 MD number	JOG_AND_POS_MAX_JERK Axial jerk	
Default: 1000.0	Min. input limit: 0.0	Max. input limit: ***
Change valid after Power On	Protection level: 2/7	Unit: 0.1 m/s^3, rev/s^3
Data type: DOUBLE	Valid as from SW version: 3	
Meaning:	The jerk limiting value limits the change of the axis acceleration in the modes JOG, REF.	
MD not applicable to	path interpolation and error conditions resulting in quick stop	
Related to	MD: JOG_AND_POS_JERK_ENABLE (default setting of axial jerk limiting)	

35220 MD number	ACCEL_REDUCTION_SPEED_POINT Speed for reduced acceleration	
Default: 1.0	Min. input limit: 0.0	max. Input limit: 1.0
Change valid after Power On	Protection level: 2/7	Unit: Factor
Data type: DOUBLE	Valid as from SW version:	
Meaning:	The machine data defines the threshold speed/threshold velocity for the spindle/axes from which the acceleration reduction is to start. The defined maximum speed/velocity is used as the reference. The threshold value depends on the maximum values as a percentage. This application is recommended for stepper motor axes. Example: MD: ACCEL_REDUCTION_SPEED_POINT = 0,7, the maximum speed is 3,000 rpm. At v_{ein}= 2,100 rpm, the acceleration reduction starts, i.e. the maximum acceleration capability is utilized in the speed range 0...2,099.99 rpm. From 2,100 rpm to the maximum speed, a reduced acceleration is used.	
Related to	MD 32000: MAX_AX_VELO (maximum axis velocity) MD 35130: GEAR_STEP_MAX_VELO_LIMIT (maximum speed of gear stage) MD 35230: ACCEL_REDUCTION_FACTOR (reduced acceleration)	

35230 MD number	ACCEL_REDUCTION_FACTOR Reduced acceleration		
Default: 0.0	Min. input limit: 0.0	Max. input limit: 1.0	
Change valid after Power On	Protection level: 2/7	Unit: Factor	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The machine data contains the factor by which the acceleration of the spindle/axes is reduced with reference to the maximum speed/velocity. From the threshold speed/velocity determined based on MD:ACCEL_REDUCTION_SPEED_POINT up to the maximum speed/velocity, the acceleration is reduced by the factor. This application is recommended for stepper motor axes. Example: a= 10 rev/s^2, v_{ein}= 2100 rpm, MD: ACCEL_REDUCTION_FACTOR = 0.3. The acceleration/deceleration is carried out in the speed range 0...2,099.99 rpm with an acceleration of 10 rev/s^2. From the speed 2,100 rpm onwards, the acceleration is reduced up to the maximum speed of 10 rev/s^2 down to 7 degrees/s^2.		
MD not applicable to	errors resulting in quick stop		
Related to	MD: MAX_AX_ACCEL (axis acceleration) MD: GEAR_STEP_SPEEDCTRL_ACCEL (acceleration in speed control mode) MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode) MD: ACCEL_REDUCTION_SPEED_POINT (speed for reduced acceleration)		

35230 MD number	ACCEL_TYPE_DRIVE Acceleration reduction ON/OFF		
Default for the axes: **802S:** (1, 1, 1, 0) **802C:** (0, 0, 0, 0)	Min. input limit: 0	Max. input limit: 1	
Change valid after Power On	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	Default setting of acceleration behavior for all traversing movements **0:** No acceleration reduction **1:** Acceleration reduction active Application of acceleration reduction recommended for stepper motor axes		
MD applicable to	JOG_AND_POS_JERK_ENABLE = 1		
Related to	MD: JOG_AND_POS_JERK_ENABLE MD: ACCEL_REDUCTION_TYPE MD: ACCEL_REDUCTION_FACTOR MD: ACCEL_REDUCTION_SPEED_POINT		

Setting data

41110 SD number	JOG_SET_VELO JOG velocity for linear axes (for G94)		
Default: 0	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	Protection level:	Unit: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Value unequal to 0: The entered velocity is used when linear axes are traversed in JOG mode. The axis velocity acts for – continuous traversing – incremental traversing (INC1, ...) – handwheel traversing The entered value is valid for all linear axes together and may not exceed the maximum permissible axis velocity (MD: MAX_AX_VELO). Value = 0: If “0” is entered in the setting data, the MD:JOG_VELO “Conventional axis velocity” acts as feed for the linear axes in JOG mode. It is possible to assign each axis its own JOG velocity (axis MD).		
Application example(s)	The operator can use these setting data to assign an application–specific JOG velocity.		
Related to	Axis–specific MD: JOG_VELO (conventional axis velocity) Axis–specifc MD: MAX_AX_VELO (maximum axis velocity)		

41200 SD number	JOG_SPIND_SET_VELO JOG velocity for spindle		
Default: 0	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	Protection level:	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Value unequal to 0: The entered velocity acts for spindles in JOG mode when traversing them manually using the traversing keys plus or minus. The velocity acts for – continuous traversing – incremental traversing (INC1, ...) – handwheel traversing The entered value applies to all spindles together and may not exceed the maximum permissible velocity (MD: MAX_AX_VELO). Value = 0: If “0” is entered in the setting data, the MD: JOG_VELO (conventional axis velocity) acts as JOG velocity. These machine data can be used to assign each axis its own JOG velocity (axis MD). When the spindle is traversed in JOG mode, the maximum velocities of the active gear stage (MD: GEAR_STEP_VELO_LIMIT) are used.		
SD not applicable to	axes		
Application example(s)	The operator can use these setting data to set an application–specific JOG velocity for spindles.		
Related to	Axis MD: JOG_VELO (conventional axis velocity) MD: GEAR_STEP_MAX_VELO_LIMIT (maximum speed of gear stages)		

4.8 Signal Description

VB19001003 and VB19001004 Interface signals	Axis Number for handwheel 1 or 2 Signal(s) from NC (MMC –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal meaning	The operator can assign each handwheel an axis directly on the operator panel. To this aim, he defines the desired axis (e.g., X). The axis number relating to the axis with the information “Machine/axis” (axis IS) is made available to the axis from the PLC basic program as MMC interface signals. The PLC basic program sets thus the interface signal “Activate handwheel” for the defined axis. Depending on the MMC interface signal “Machine axis”, the interface to the machine axis or to the axis in WCS is used. The following is applicable when assigning the axis name to the axis number: • IS ”Axis” = 1; i.e. axis: X IS “Axis” = 2; i.e. axis: Y IS “Axis” = 3; i.e. axis: Z The following coding is used for the axis number:		
Related to	IS ”Axis“ (V19001003.7 or V19001004.7) IS ”Activate handwheel“		

Bit 1	Bit 0	Axis Number
0	0	–
0	1	1
1	0	(2)
1	1	3

V19001003.7 and V19001004.7 Interface signal	Axis (for Handwheel 1 or 2) Signal(s) from NC (MMC –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	The operator has assigned the handwheel (1, 2) directly on the operator panel. This axis is a machine axis (MCS). For more information refer to IS “Axis number”.		
Signal status 0 or edge change 1 ——> 0	The operator has assigned the handwheel (1, 2) an axis directly on the operator panel. This axis is a WCS axis. For more information refer to IS “Axis number”.		
Related to	IS ”Axis number“ VB19001003 ff) IS ”Handwheel selected“ (V19001003.6)		

V32001000.0 to 1 V32001004.0 to 1 V32001008.0 to 1 Interface signal	Activate handwheel (1 to 2) for axis in WCS Signal(s) to channel (PLC –> NCK)	
Edge evaluation: nein	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	These PLC interface signals are used to define whether the axis is assigned to handwheel 1 or 2 or to no handwheel. Only one handwheel each can be assigned to one axis at a time. If several interface signals ”Activate handwheel“ are set, the priority is ’Handwheel 1’ before ’Handwheel 2’. If the assignment is active, the axis can either be traversed in JOG mode using the handwheel or a DRF offset can be generated in AUTOMATIC or MDA mode. **Note:** Using handwheel 1 to 2, two axes can be traversed simultaneously.	
Signal status 0 or edge change 1 ——> 0	This axis is not assigned handwheel 1 or 2.	
Application example(s)	The interface signal can be used to interlock axis control from the PLC user program by turning the handwheel.	
Related to	IS ”Handwheel active“ for axis	

V32001000.4 V32001004.4 V32001008.4 Interface signal	Traversing key for WCS axis blocked Signal(s) to channel (PLC –> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	The traversing keys plus and minus have no effect for the axis in question. For example, traversing the axis in JOG mode using the traversing keys on the MCP is not possible. If traversing key blocking is activated while the axes are traversed, the axis is stopped.	
Signal status 0	The traversing keys plus and minus are enabled.	
Application example(s)	It is thus possible to interlock traversing of the axis in JOG mode via the traversing keys from the PLC user program, depending on the operating condition.	
Related to	IS ”Traversing key plus” and ”Traversing key minus” for axis	

V32001000.5 V32001004.5 V32001008.5 Interface signal	Rapid traverse override for WCS axis Signal(s) to channel (PLC –> NCK)	
Edge evaluation: nein	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	When the PLC interface signal “Rapid traverse override” is provided at the same time when the “Traversing key plus” or “Traversing key minus” are pressed, the respective axis will traverse with rapid traverse. The rapid traverse velocity is defined by the machine data JOG_VELO_RAPID. The rapid traverse override is active in JOG mode with the following variants: – continuous traversing – incremental traversing With rapid traverse override active, the velocity can be controlled by the rapid traverse override switch.	
Signal status 0 or edge change 1 ——> 0	The axis traverses with the set JOG velocity (SD: JOG_SET_VELO or MD: JOG_VELO).	
Signal not applicable to	– AUTOMATIC mode and MDA – Reference–point approach (JOG mode)	
Related to	IS ”Traversing key plus“ and ”Traversing key minus“ for axis	

V32001000.7 and .6 V32001004.7 and .6 V32001008.7 and .6 Interface signal	Traversing keys plus and minus for WCS axis Signal(s) to channel (PLC –> NCK)		
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	In JOG mode, the selected axis can be traversed in both directions by means of the traversing keys plus and minus. **Incremental traversing** On signal status 1, the axis starts traversing by the set increment. If the signal changes to 0 status before the increment is traversed, the traversing movement is interrupted. If the signal status is “1” again, the traversing movement is continued. As long as the increment is traversed completely, the traversing movement of the axis can be stopped and continued several times as described above. **Continuous traversing** If no INC dimension is selected, the axis traverses as long as the traversing key is pressed. If both traversing signals (plus and minus) are set at the same time, no traversing movement is carried out or the traversing movement is aborted. The PLC interface signal ”Traversing key blocked“ can be used to block the effect of the traversing keys for each axis individually.		
Signal status 0 or edge change 1 ——> 0			
Signal not applicable to	in AUTOMATIC mode and MDA		
Related to	IS ”Traversing key for axes blocked“		

V32001000.0 to .3, .6 V32001004.0 to .3, .6 V32001008.0 to .3, .6 Interface signal	Machine function for WCS axis INC1, INC10, INC100, INC 1000, continuous Signal(s) to channel (PLC –> NCK)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	These interface signals are used to define how many increments are covered by the axis when the direction key is pressed or the handwheel is turned, or it is continuous mode. When doing this, JOG mode must be active. Once the selected machine function is active, this is reported to the PLC interface (IS ”Active machine function INC1; ...”). If several machine function signals (INC1, INC...) are selected on the interface, no machine function is activated by the control system.		
Signal status 0 or edge change 1 ——> 0	The respective machine function is not selected. If an axis is just traversing an increment, the movement is also aborted by deselecting or changing the machine function.		

V33001000.0 and .1 V33001004.0 and .1 V33001008.0 and .1 Interface signal	Handwheel active (1 to 2) for WCS axis Signal(s) from channel (NCK –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	These PLC–interface signals are used to report whether the axis is assigned to handwheel 1 or 2 or to none handwheel. At this moment, one axis can be assigned only one handwheel each. If several interface signals “Activate handwheel” are set, the priority is: ’Handwheel 1’ before ’Handwheel 2’. If the assignment is active, the axis can be traversed with the handwheel in JOG mode.		
Signal status 0 or edge change 1 ——> 0	This axis is not assigned handwheel 1 or 2.		
Related to	IS ”Activate handwheel”		

V33001000.7 and .6 V33001004.7 and .6 V33001008.7 and .6 Interface signal	Traversing command plus and minus for WCS axis Signal(s) from channel (NCK –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	A traversing motion is desired in the respective axis direction. Depending on the operating mode, the traversing command is triggered in different ways: – JOG mode: by means of the traversing key plus or minus – REF mode: by means of the traversing key used for reference–point approach – AUT/MDA mode: A program block that contains a coordinate value for the respective axis is executed.		
Signal status 0 or edge change 1 ——> 0	At the moment, no traversing request is provided in the axis direction in question, or a traversing motion is completed. • JOG mode: – The traversing command is reset depending on the interface signal “Traversing keys plus and minus”. – when traversing with the handwheel. • REF mode: when the reference–point is reached. • AUT/MDA mode: – The program block is executed (and the following block does not contain a coordinate value for the axis in question). – Abortion by RESET, etc. – IS “Axis blocking” provided		
Application example(s)	To release axis clamping (e.g. rotary tables). **Note:** If the clamping is only released with the traversing command, no path–control mode is possible with these axes.		
Related to	IS ”Traversing key plus“ and “Traversing key minus” for WCS axis		

V33001001.0 to .3 V33001005.0 to .3 V33001009.0 to .3 Interface signal	Active machine function for WCS axis INC1, ..., INC 1000, continuous Signal(s) from channel (NCK –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	The PLC interface receives a check–back signal with the information which machine function is active for the axes in JOG mode. Depending on which machine function is active, the response when pressing the traversing key or turning the handwheel is different.		
Signal status 0 or edge change 1 ——> 0	The respective machine function is not active.		
Related to	IS ”Machine function INC1,..., INC1000” for WCS axis		

4.8.1 Overview of Signals to Axis/Spindle (Machine Axis)

	Signals to Axis/Spindle							
VB	**Bit 7**	**Bit 6**	**Bit 5**	**Bit 4**	**Bit 3**	**Bit 2**	**Bit 1**	**Bit 0**
380x0004	**Traversing keys**		**Rapid Traverse Override**		**Feed Stop/ Spindle Stop**	**Activate handwheel**		
	plus	**minus**					**2**	**1**
	Machine function							
380x0005		**conti-nous**			**1000 INC**	**100 INC**	**10 INC**	**1 INC**

4.8.2 Description of Signals to Axis/Spindle (Machine Axis)

V380x0004.0 and .1 Interface signal	Activate handwheel (1 to 2) Signal(s) to axis/spindle (PLC –> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	These PLC–interface signals are used to define whether this axis is assigned to handwheel 1 or 2 or to none. One axis can be assigned only one handwheel at a time. If several interface signals ”Activate handwheel“ are set, the priority is: ’Handwheel 1’ before ’Handwheel 2”. If the assignment is active, the axis can be traversed with the handwheel in JOG mode or a DRF offset generated in AUTOMATIC or MDA mode.	
Signal status 0 or edge change 1 ——> 0	This axis is handwheel 1 or 2 not assigned.	
Application example(s)	This interface signal can be used to interlock axis control by turning the handwheel from the PLC user program.	
Related to	IS ”Handwheel active“	

V380x0004.5 Interface signal	Rapid traverse override Signal(s) to axis/spindle (PLC –> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1	If the PLC interface signal “Rapid traverse override” is provided together with “Traversing key plus” or “Traversing key minus”, the called axis traverses with rapid traverse. The rapid traverse velocity is set in the machine data JOG_VELO_RAPID. Rapid traverse override is active in JOG mode in the following cases: – continuous traversing – incremental traversing When rapid traverse override is active, the velocity can be controlled by means of the axis feed override switch.	
Signal status 0 or edge change 1 ——> 0	The axis traverses with the set JOG velocity (SD: JOG_SET_VELO or MD: JOG_VELO).	
Signal not applicable to	– AUTOMATIC mode and MDA – Reference–point approach (JOG mode)	
Related to	IS ”Traversing key plus” and ”Traversing key minus” IS ”Axis feed/spindle override”	

V380x0004.7 and .6 Interface signal	Traversing keys plus and minus Signal(s) to axis/spindle (PLC –> NCK)		
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	In JOG mode, the selected axis can be traversed in both directions by means of the traversing keys. **Incremental traversing** On signal status 1, the axis starts traversing by the set increment. If the signal changes to 0 before the increment is traversed, the traversing movement is interrupted. If the signal status is “1” again, the traversing movement is continued. As long as the increment is traversed completely, the traversing movement of the axis can be stopped and continued several times as described above. **Continuous traversing** If no INC dimension is selected, the axis will traverse as long as the key remains pressed. If both traversing signals (plus and minus) are set at the same time, no traversing movement is carried out or the traversing movement is aborted. The PLC interface signal ”Traversing key blokked” can be used to block the effect of the traversing keys for each axis separately.		
Signal status 0 or edge change 1 ——> 0			
Signal not applicable to	AUTOMATIC mode and MDA		
Application example(s)	The axis cannot be traversed in JOG mode if it is already traversed via the channel–specific PLC interface (as an axis). Alarm 20062 is output.		
Special cases, ...	Pitch axes		
Related to	IS ”Traversing keys plus and minus for WCS axes” IS ”Traversing keys blocked“		

V380x0005.0 to .3, .6 Interface signal	Machine function INC1, INC10, INC100, INC1000, continuous Signal(s) to axis/spindle (PLC –> NCK)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	These interface signals are used to define how many increments are covered by the axis when the traversing keys are pressed or per latched position when the handwheel is turned, or it is continous mode. JOG mode must be active. As soon as the selected machine function is active, this is reported to the PLC interface (IS “Active machine function INC1; ...”). If several machine function signals (INC1, INC... or ”Continuous traversing“) are selected on the interface at the same time, no machine function is activated by the control system.		
Signal status 0 or edge change 1 ——> 0	The respective machine function is not selected. If an axis is just traversing an increment, the motion is also aborted by deselecting or changing the machine function.		
Related to	IS ”Active machine function INC1, ...”		

4.8.3 Overview of Signals from Axis/Spindle (Machine Axis)

	Signals from Axis/Spindle							
VB	**Bit 7**	**Bit 6**	**Bit 5**	**Bit 4**	**Bit 3**	**Bit 2**	**Bit 1**	**Bit 0**
390x0004	**Traversing command**					**Handwheel active**		
	plus	**minus**					**2**	**1**
390x0005	**Active machine function**							
		continu-ous			**1000 INC**	**100 INC**	**10 INC**	**1 INC**

4.8.4 Description of Signals from Axis/Spindle (Machine Axis)

V390x0004.0 and .1 Interface signal	Handwheel active (1 to) Signal(s) from axis/spindle (NCK –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	These PLC interface signals are used to check back whether this axis is assigned to handwheel 1 or 2 or to none handwheel. One axis can be assigned only one handwheel each at a time. If several interface signals ”Activate handwheel” are set, the priority is ’Handwheel 1’ before ’Handwheel 2’. If the assignment is active, the axis can be traversed with the handwheel in JOG mode.		
Signal status 0 or edge change 1 ——> 0	This axis is **not** assigned handwheel 1 or 2.		
Related to	IS ”Activate handwheel“ IS ”Handwheel selected”		

V390x0004.7 and .6 Interface signal	Traversing command plus and minus” Signal(s) from axis/spindle (NCK –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	A traversing movement is desired in the respective axis direction. Depending on the operating mode, the traversing command is triggered in different ways: – JOG mode: by means of the traversing key plus or minus – REF mode: by means of the traversing key used for reference–point approach – AUT/MDA mode: A program block that contains a coordinate value for the respective axis is executed.		
Signal status 0 or edge change 1 ——> 0	At the moment, no traversing request is provided in the axis direction in question, or a traversing motion is completed. • JOG mode: – The traversing command is reset depending on the interface signal “Traversing keys plus and minus”. – when traversing with the handwheel. • AUT/MDA mode: – The program block is executed (and the following block does not contain a coordinate value for the axis in question). – Abortion by RESET, etc. – IS “Axis blocking” provided		
Application example(s)	**Note:** If the clamping is only released with the traversing command, continuous–path control is not possible for these axes.		
Related to	IS ”Traversing key plus“ and ”Traversing key minus”		

V390x0005.0 to .3, .6 Interface signal	Active machine function INC1, ...INC 1000, continuous Signal(s) from axis/spindle (NCK –> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ——> 1	A check–back signal with the information which machine function is active for the axes in JOG mode is provided to the PLC interface. Depending on the active machine function, the response when pressing the traversing key or turning the handwheel is different.		
Signal status 0 or edge change 1 ——> 0	The respective machine function is not active.		
Related to	IS ”Machine function INC1,...,		

MD–Nummer			
Standardvorbesetzung:	min. Eingabegrenze:	max. Eingabegrenze:	
Änderung gültig nach	Schutzstufe:	Einheit:	
Datentype:	gültig ab SW–Stand:		
Bedeutung:			
MD irrelevant bei			
Bild ????.???			
Anwendungsbeispiel(e)			
Sonderfälle, Fehler, ...			
korrespondierend mit			
weiterführende Literatur			

notice

Program Mode

5

Brief description

Program mode means that part programs or part program blocks are executed in AUTOMATIC mode or MDA. During block execution, the program can be controlled by PLC interface signals.

Channel

A channel is a unit in which a part program can be executed.

The system assigns the channel an interpolator with the associated program processing. The channel is assigned a certain operating mode.

5.1 Operating Modes

The following operating modes are possible:

AUTOMATIC

Automatic execution of part programs

MDA

1 program block can be executed.

JOG

Traversing of the axes by manual operation via handwheel or traversing keys; channel–specific signals and interlocks are ignored.

Activation

The desired operating mode is activated via the interface signals in VB30000000. If several operating modes are selected at the same time, the following priority applies:

- JOG (high priority)
- MDA
- AUTOMATIC (lower priority)

Check–back signals

The active operating mode is indicated via the interface signals in VB 31000000.

Check–back signals

In JOG mode, the following machine functions can be selected:

- REF (reference–point approach)

The desired machine function is activated in IS VB30000001 (IS = interface signal).
The active machine function is indicated in IS VB31000001.

Stop

The IS ”NC Stop” (V32000007.3), IS ”NC Stop axes and spindles“ (V32000007.4) or ”NC Stop at block end“ (V32000007.2) can be used to provide a stop signal. Depending on the number of stop signals, either only the axes or, in addition, the spindle or axes are stopped at the end of the block.

RESET

The IS ”Reset” (V30000000.7) aborts the active part program.

The following activities are carried out after the IS “Reset” is triggered off:

- Part program preparation is stopped immediately.
- Axes and spindles are stopped.
- The auxiliary functions of the currently active block, which are not yet output at this moment, are not output.
- The block pointers are reset to the beginning of the respective part program.
- All Reset alarms are deleted from the display.
- The Reset operation is completed with setting the IS “Channel status RESET” (V33000003.7).

Ready for operation

Readiness for operation is signalled by the IS “Ready” (V 31000000.3).

5.1.1 Mode Change

General

Mode change is requested and activated via the interface.

Note

The control system will only change its mode if “Channel status active” is no longer present.

Mode change is only allowed if the machine stops. In channel status “Reset” (IS V33000003.7, e.g. after pressing the Reset key), it is possible to change from one mode to another.

For example, if you leave AUTO to change for JOG, you must either return to AUTO or press Reset. This makes a change AUTO–JOG–MDA impossible. The same applies to MDA from which you cannot change nor directly, neither indirectly to AUTO if the control system is not in Reset condition.

The mode changes possible depending on the current operating mode and the channel status are listed in the Table below.

Table 5-1 Mode changes

	from	AUTOMATIC		JOG			MDA	
					Before AUTO	Before MDA		
to		Reset	Interr.	Reset	Interr.	Interr.	Reset	Interr.
AUTOMATIC				X	X		X	
JOG		X	X				X	X
MDA		X		X		X		

Possible mode changes are marked with an ”X”.

Mode change errors

If mode change request has been rejected by the system, a respective error message is output. This error message can be deleted without changing the channel status.

Mode change blocked

The IS “Mode change blocked“ (V30000000.4) can be used to prevent mode change. Already the mode change request is suppressed.

5.1.2 Possible Functions in the Individual Modes

Overview of functions

Which function in which operating mode and which operating condition can be selected is to be seen in the following Table.

Table 5-2 Possible functions in the individual operating modes

	Channel in Reset AUTOMATIC status	Channel interrupted	Channel active	Channel in Reset JOG status	Channel active	Channel interrupted JOG during AUTO interr.	Channel active	Channel interrupted JOG during MDA interr.	Channel active	Channel in Reset MDA status	Channel interrupted	Channel active	Channel active JOG in MDA during MDA interr.	Channel active JOG in MDA
Functionalities														
Loading a part program from externals via “Services”	sb	sb		sb		sb		sb	sb	sb	sb			
Executing a part program/block	s	s	b							s	s	b		
Block search	s	s	b											
Reference–point approach via part program command			sb									sb		
s: Function cannot be started in this status b: Function can be executed in this status														

5.1.3 Monitoring Functions in the Individual Operating Modes

Overview of monitoring functions

In the individual operating modes, different monitoring functions are active.
Which monitoring functions are active in which operating condition is to be seen in the following Table.

Table 5-3 Monitoring functions in the individual operating modes

	Channel in Reset AUTOMATIC status	Channel interrupted	Channel active	Channel in Reset JOG status	Channel active	Channel interrupted JOG during AUTO interr.	Channel active	Channel interrupted JOG during MDA interr.	Channel active	Channel in Reset MDA status	Channel interrupted	Channel active	Channel active JOG in MDA during MDA interr.	Channel active JOG in MDA
Axis–specific monitoring functions / monitoring functions active when positioning the spindle														
SW limit switches +			x		x		x		x			x	x	x
SW limit switches -			x		x		x		x			x	x	x
HW limit switches +	x	x	x	x	x	x	x	x	x	x	x	x	x	x
HW limit switches -	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Exact stop coarse/fine	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Clamping tolerance	x	x	x	x	x	x	x	x	x	x	x	x	x	x
DAC limiting	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Contour monitoring			x		x		x		x			x	x	x
Spindle–specific monitoring functions														
Speed limit exceeded			x		x		x		x			x		x
Spindle on stop	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Spindle synchronized			x		x		x		x			x		x
Speed within set range			x											
Max. permissible speed			x		x		x		x			x		x
Encoder limit frequency			x		x		x		x			x		x
x: Monitoring is active in this status														

5.1.4 Interlocks in the Individual Modes

Overview of interlocks

In the individual operating modes, different interlocks can be active.

Which interlocks can be activated in which operating mode and which operating condition is to be seen in the Table below:

	Channel in Reset AUTOMATIC status	Channel interrupted	Channel active	Channel in Reset JOG mode	Channel active	Channel interrupted JOG during AUTO interr.	Channel active	Channel interrupted JOG during MDA interr.	Channel active	Channel in MDA Reset status	Channel interrupted	Channel active	Channel active JOG in MDA during MDA interr.	Channel active JOG in MDA
General interlocks														
Ready	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Mode change blocked	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Channel–specific interlocks														
Feed Stop			x		x		x		x			x	x	x
NC Start inhibited	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Read–in disable	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Axis–specific interlocks														
Spindle blocked	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Servo disable	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Axis blocked	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Spindle–specific interlocks														
Servo disable	x	x	x	x	x	x	x	x	x	x	x	x	x	x
Spindle blocked	x	x	x	x	x	x	x	x	x	x	x	x	x	x
x: Interlock can be activated in this status														

5.2 Program Test

Objective

To test or try a new part program, various control functions are used. These functions are designed such that the hazards to the machine during the test phase and the time required for testing are considerably reduced. It is possible to activate several program test functions at the same time.

The following test options are described here:

- Program execution without axis movements (PRT program test)
- Program execution in single block mode (SBL)
- Program execution with dry run feed (DRY)
- Processing of certain program sections using block search
- Skipping of certain program parts (SKP)

5.2.1 Program Execution Without Axis Movements (Program Test)

Functionality

The part program can be started and executed including auxiliary function outputs and dwell times via the IS “NC Start” if the Program Test function is active. The only difference to real processing is that the axes/spindle are merely simulated. The safety function “Software limit switch” remains active.

The only difference to normal program execution is that an internal Axis/Spindle Blocked signal is output for all axes, i. e. the machine axes do not move, and the actual values are internally generated from the setpoints which are not output. The programmed velocities remain unchanged. That means that the position and velocity specifications on the operator interface exactly correspond to those of normal part program execution.

The position control is not interrupted during this process so that it is not necessary to reference the axes after switching off.

Application

The user can use this function to check the programmed axis position and the auxiliary outputs of a part program.

Selection

This function is selected via the operator interface in the Program Control menu. With the selection, the IS “Program test selected” (V17000001.7) is set. The function is thus not yet activated.

Activation

The function is activated via the IS “Activate program test” (V32000001.7).

Display

As a feedback information that program test is active, “PRT” is displayed in the status line on the operator interface and the IS “Program test active” (V33000001.7) is set in the PLC.

Note

Program execution without axis movements can also be activated together with the function “Dry run feed”.

5.2.2 Program Execution in Single Block Mode

Functionality

The part program can be started via the IS ”NC Start” (V32000007.1).
However, the part program execution stops after each program block if the function “Single block” is active.
The program status changes to “Program status stopped”. The channel status remains active.

Execution of the next part program block is started with “NC Start”.

Single block type

The following single block types are differed:

- Action Single Block type (SBL 1)
  With this single block type, all blocks that trigger actions (traversing movements, auxiliary function outputs etc.) are executed separately. If tool radius compensation is switched on (G41,G42), the program stops after each intermediate block inserted by the control system. In case of arithmetic blocks, however, the program execution does not stop, since arithmetic blocks do not trigger actions.
- Decoding Single Block (SBL 2)
  With this single block type, **all** blocks of the part program (including pure arithmetic blocks without traversing movements) are executed one after another by NC Start.

Action Single Block (SBL1) is the default setting after switching on.

Caution

- Single Block in a series of G33 blocks is only active if Dry Run Feed is selected.
- Arithmetic blocks are not processed in Single Block mode (only in Decoding Single Block mode –SBL2).

Application

The user can use this function to execute a part program block by block in order to check the individual steps of processing. If he has found the executed part program part correct, he can request the next block. Changing to the next part program block is carried out by “NC Start”.

Selection

Single Block mode is selected by means of the SBL key on the machine control panel. With the selection, the IS “Single block selected” (V00000001.2) is set. The function is thus not yet activated.
The preselection whether type “SBL1” or “SBL2” is carried out on the operator interface in the Program Control menu.

Activation

This function is activated via the IS ”Activate single block“ (V32000000.4).

Display

As a feedback information that Single Block mode is active, “SBL1” or “SBL2” is displayed in the respective field on the operator interface. Once the part program execution has executed a part program block due to Single Block mode, the IS “Program status interrupted” (V33000003.3) is set.

5.2.3 Program Execution With Dry Run Feed

Functionality

The part program can be started via the IS ”NC Start” (V32000007.1). With the function activated, the traversing velocities programmed in conjunction with G1, G2, G3, G5 are replaced by the feedrate value stored in SD: DRY_RUN_FEED. The dry run feed value is also applicable in program blocks with G95, instead of the programmed revolutional feedrate.

Danger

When the Dry Run Feed function is active, workpiece machining is not allowed, since the cutting speed of the tools could be exceeded or the workpiece or machine tool could be destroyed due to the modified feedrate values.

Selection

The operation with dry run feed is selected on the operator interface in the Program Control menu. Together with the selection, the IS “Dry run feed selected” (V17000000.7) is set. In addition, the desired value for dry run feed must be entered in the Setting Data menu. The function is thus not yet activated.

Activation

The function is activated via the IS “Activate dry run feed” (V32000000.4).

Display

As a feedback information that dry run feed is active “DRY” is displayed in the status line on the operator interface.

5.3 Processing of Certain Program Parts

Functionality

If only a certain program section is to be checked, the Block Search function can be used to jump to the beginning of the program section concerned.
After block search, the program can be started via the IS ”NC Start” (to be provided 2x) (V32000007.1).

Selection, activation

Block Search is selected and activated on the operator interface in AUTOMATIC mode.

Feedback

As a feedback information that Block Search is active the IS ”Block search active“ (V33000001.4) is set.

Note

For more information on the Block Search function see

Documentation: “Operation and Programming“

5.3.1 Skipping Certain Part Program Blocks

Functionality

When testing or starting up new programs, it is useful to skip certain part program blocks in the program execution.

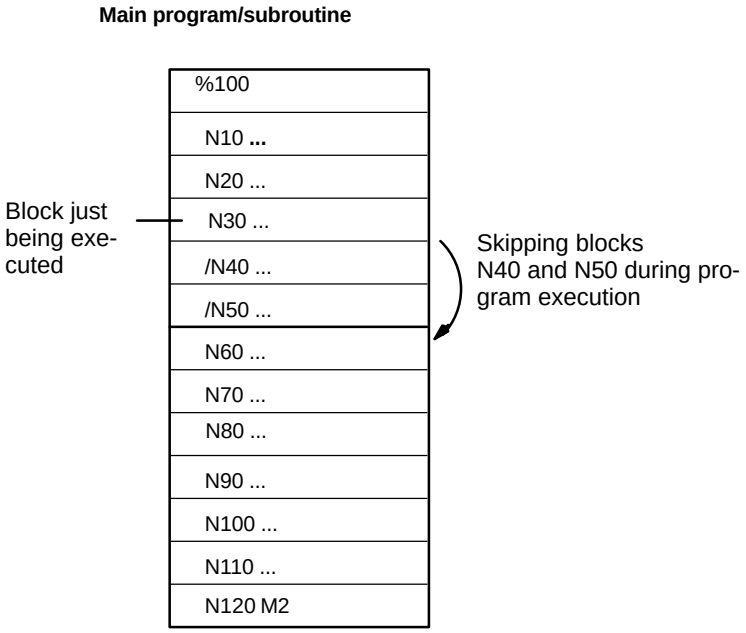


Fig. 5-1 Skipping part program blocks

Selection

The Block Skip function is selected on the operator interface in the Program Control menu. Together with the selection, the IS “Skip block” (V17000002.0) is set. In addition, a slash ”/ ” must be set in front of the blocks (see Fig. 5-1). The function is thus not yet activated.

Activation

The function is activated via the IS ”Activate block skip” (V32000002.0).

Display

As a feedback information that the Block Skip function is active “SKP” is displayed in the status line on the operator interface.

5.4 Executing a Part Program

Definition

Program mode means that either a part program is executed in AUTOMATIC mode or a program block is executed in MDA mode.

Control

During program mode, the program can be controlled by interface signals from the PLC. Controlling is carried out via mode–specific or channel–specific interface signals.

Feedbacks of the control

The channel informs the PLC of its current program mode status via interface signals.

5.4.1 Part Program Selection

Channel status

A part program can only be selected if the channel is in Reset status.

5.4.2 Starting the Part Program or Part Program Block

START command, channel status

The channel–specific IS ”NC Start” (V32000007.1), which is commonly controlled from the MCP key “NC Start”, starts program execution.

The START command is only executed in AUTOMATIC and MDA mode. The channel must be either in the status ”Channel status Reset” (V33000003.7) or ”Channel status interrupted“ (V33000003.6).

Required signal states

The selected part program can now be enabled for execution using the START command. The following enable signals can be used:

- IS ”Ready” must be set (V31000000.3).
- IS ”Activate program test” must not be set (V32000001.7).
- IS ”NC Start inhibited“ must not be set (V32000007.0).
- IS ”NC Stop at block end” must not be set (V32000007.2).
- IS ”NC Stop” must not be set (V32000007.3).
- IS ”NC Stop axes plus spindle“ must not be set (V32000007.4).
- IS ”EMERGENCY STOP” must not be set (V27000000.1).
- Axis or NCK alarm must not be present.

Command execution

The part program or part program block, resp., is executed automatically, and the IS ”Channel status active“ (V33000003.5) and the IS ”Program status running“ (V33000003.0) are set. The program is executed as long as the program end is reached or the channel is interrupted or aborted by a STOP or RESET command.

Alarms

The START command does not come into effect if the precondition is not fulfilled. In this case, one of the following alarms is output: 10200, 10202 , 10203

5.4.3 Interrupting a Part Program

Channel status

The STOP command can only be executed if the channel in question has the status “Channel active” (V33000003.5).

STOP commands

There are different commands that stop program execution and set the channel status to “Interrupted”. These commands are:

- IS ”NC Stop at block end“ (V32000007.2)
- IS ”NC Stop” (V32000007.3)
- IS ”NC Stop axes plus spindle“ (V32000007.4)
- IS ”Single block“ (V32000000.4)
- Programming command ”M0” or ”M1”

Command execution

After the STOP command has been executed, the IS ”Program status interrupted” (V33000003.3) is set. The interrupted part program can be continued from the breakpoint by another START command.

Generally, the following actions are carried out after STOP command:

- Stopping part program execution at the end of the next block (with NC Stop at block end, M0/M1 or single block), the remaining STOP commands will stop program execution immediately.
- Auxiliary functions of the current block which are not yet output at this moment will not be output.
- The axes are stopped with successive stop of part program execution.
- The block pointer stops on the breakpoint.

5.4.4 RESET Command

Channel status

The RESET command can be executed in any channel status. There is no command that can cancel this command.

Reset commands

The following Reset command can be used:

IS ”Reset” (V3000000.7)

Command execution

RESET commands can be used to abort an active part program or part program block (in MDA).
When the Reset command is executed, the IS ”Channel status Reset” (V33000003.7) is set.
The part program can no longer be continued from the breakpoint. All axes in the channel are in exact stop status.

After the Reset command, the following actions are carried out:

- Part program preparation is stopped immediately.
- The axes and the spindle (if any) are decelerated.
- Any auxiliary functions of the current block, which are not yet output at this moment, will not be output any more.
- The block pointer is reset to the beginning of the part program.
- All alarms are deleted from the display if they are not POWER ON alarms.

5.4.5 Program Control

The user can control program execution via the operator interface.

Selection

The Program Control softkey can be used to select certain functions on the operator interface; some signals affect interface signals of the PLC. These interface signals are merely intended as selection signals from the operator interface. They are not intended to activate the selected function.

Activation

To bring the selected functions into effect, the respective signal states have to be transferred to another range of the data block. In case of controlling from the PLC, these signals must be set directly.

Feedback

Some activated functions have a check–back signal.

Table 5-4 Program control

Function	Selection Signal	Activation Signal	Check–Back Signal
SKP Skippable block	V17000001.0	V32000002.0	
DRY Dry run feed	V17000000.6	V32000000.6	
ROV Rapid traverse override	V17000001.3	V32000006.6	
Preselection: SBL1 – single block type 1 SBL2 – single block type 2 Key: Single block	 – – V00000001.2	 – – V32000000.4	
M1 Programmed stop	V17000000.5	V32000000.5	V33000000.5
PRT Program test	V17000000.7	V32000001.7	V33000001.7

5.4.6 Program Status

The status of the selected program for the channel is displayed on the interface. The program status is only displayed in AUTOMATIC mode and MDA. In the remaining operating modes, the program status is either aborted or interrupted.

Program states

The following program states are possible:

- IS ”Program status aborted“ (V33000003.4)
- IS ”Program status interrupted“ (V33000003.3)
- IS ”Program status stopped“ (V33000003.2)
- IS ”Program status waiting“ (V33000003.1)
- IS ”Program status running“ (V33000003.0)

Effects of commands/signals

The program status can be controlled either by activating various commands or via interface signals. The Table below shows the resulting program status (supposed program status prior to the signal -> Program status running).

Table 5-5 Effects on the program status

Commands	States of Program Execution				
	Aborted	Interrupted	Stopped	Waiting	Running
IS ”Reset”	X				
IS ”NC Stop”			X		
IS ”NC Stop at block end“			X		
IS ”NC Stop axes and spindles“			X		
IS ”Read–in disable“					X
IS ”Feed stop, channel blocked“					X
IS ”Feed stop, axis blocked”					X
Feed override = 0%					X
IS ”Spindle stop					X
M2 in block	X				
M0/M1 in block			X		
IS ”Single block“			X		
Auxiliary function output to PLC but not yet acknowledged.			X		

5.4.7 Channel Status

The current channel status is displayed on the interface. Due to the status, the PLC can trigger certain responses or interlocks, which can be configured by the manufacturer.
The channel status is displayed in all operating modes.

Channel states

The following channel states are possible:

- IS ”Channel status Reset” (V33000003.7)
- IS ”Channel status interrupted“ (V33000003.6)
- IS ”Channel status active“ (V33000003.5)

Effects of commands/signals

The channel status can be controlled either by activating various commands or via interface signals. The Table below shows the resulting channel status (supposed status prior to the signal -> channel status active).
The ”Channel status active“ is reached when a part program or part program block is executed or the axes are traversed in JOG mode.

Table 5-6 Effects on channel status

Commands	Channel Status after Command/Signal		
	Reset	Interrupted	Active
IS ”Reset”	X		
IS ”NC Stop”		X	
IS ”NC Stop at block end“		X	
IS ”NC Stop axes and spindles“		X	
IS ”Read–in disable“			X
IS ”Feed stop, channel blocked”			X
IS ”Feed stop, axes blocked”			X
Feed override = 0%			
IS ”Spindle stop“			X
M2 in block	X		
M0/M1 in block		X	
IS ”Single block“		X	
Auxiliary function output to PLC but not yet acknowledged			X

5.5 Data Description

Machine data

21000 MD number	CIRCLE_ERROR_COIS Circle end point monitoring constant		
Default: 0.01	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: mm	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	This machine data characterizes the permissible absolute circle difference. With circle programming, the radii from the programmed center point to the start point or end point are usually not the identical (the circle is “overdetermined”). The maximum permissible difference of these two radii, which are accepted without alarm, is determined by the **greater value** by the following data: - MD: CIRCLE_ERROR_COIS - Starting radius multiplied with 0.001 i.e. for smaller circles, this tolerance is a fixed value (MD: CIRCLE_ERROR_COIS), and for larger circles it is proportional to the starting radius.		
Application example	MD: CIRCLE_ERROR_COIS = 0.01 mm When these machine data and a radius $\leq$ 10 mm are used, the constant is effective; when the radius is > 10 mm, the proportional faction is effective.		

30600 MD number	FIX_POINT_POS Fixed–value positions of the axes with G75		
Default: 0	Min. input limit: 0	Max. input limit: ***	
Changes effective after Power On	Protection level: 2/7	Unit: mm, degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In these machine data, the fixed–point position that is approached when G75 is programmed is specified for each axis.		
Application example(s)	Fixed–point approach: G75 X0 (A dummy value, in this case “0”, must be specified for the axis.)		
References	”Operation and programming“		

Setting data

42100 SD number	DRY_RUN_FEED Dry run feed		
Default: 5000	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	Protection level:	Unit: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	To check a part program with refer to the distance to be traversed (without machining a workpiece), the operator can activate the function “Dry run feed” via the operator interface (Program Control softkey). In this case, instead of the programmed feedrate value, the value of this setting data is used. Rapid traverse feed values will not be modified. The dry run feed value can be entered in the setting data menu. This function is only active in AUTOMATIC mode and MDA.		
SD not applicable to	Dry Run Feed function not activated.		
Application example(s))	Checking of traversed distances for new part programs		
Special cases, errors, ...	The function may not be activated if a workpiece is to be machined. Due to the activated dry run feed, the maximum cutting speed of the workpiece could be exceeded so that workpiece and tool could be destroyed as a consequence.		

42000 SD number	THREAD_START_ANGLE Thread start angle G33		
Default: 0	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	Protection level:	Unit: degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	This setting data can be used for multiple thread cutting to set the offset of the individual threads. This SD can be modified from the part program using the command SF=... If no SF=... is programmed in the G33 block of the part program, the setting data is effective.		
References	”Operation and Programming“		

5.6 Signal Descriptions

V00000000.7 Interface signal	Selected mode JOG Signal(s) from MCP ---> PLC	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Key for JOG mode is pressed.	
Signal status 0 or edge change 1 ---> 0	Key for JOG mode is not pressed.	

V00000001.0 Interface signal	Selected machine function REF Signal(s) from MCP ---> PLC	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Key for REF is pressed.	
Signal status 0 or edge change 1 ---> 0	Key for REF is not pressed.	
References	FB ”Reference–point approach”	

V00000001.1 Interface signal	Selected mode AUTOMATIC Signal(s) MCP ---> PLC	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Key for AUTOMATIC mode is pressed.	
Signal status 0 or edge change 1 ---> 0	Key for AUTOMATIC mode is not pressed.	

V00000001.3 Interface signal	Selected mode MDA Signal(s) from MCP ---> PLC	
Edge evaluation:	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Key for MDA mode is pressed.	
Signal status 0 or edge change 1 ---> 0	Key for MDA mode is not pressed.	

V17000000.5 Interface signal	M01 selected Signal(s) from MMC ---> PLC	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Activate Program Control M1 has been selected from the operator interface. The function is thus not yet activated.	
Signal status 0 or edge change 1 ---> 0	Activate Program Control M1 has not been selected from the operator interface.	
Related to	IS ”Activate M01” IS ”M0/M1 active”	

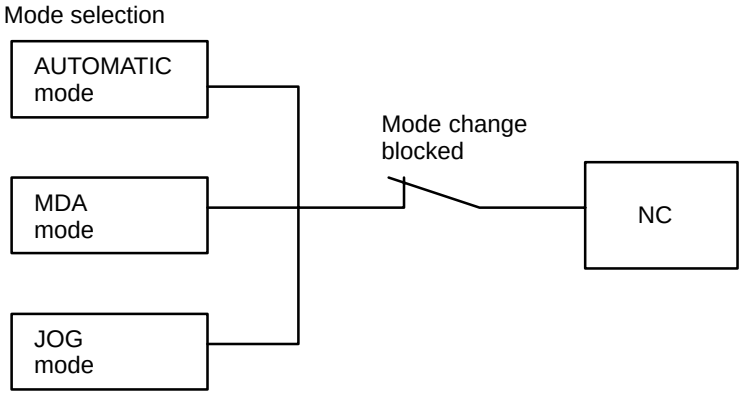
V17000001.7 Interface signal	Program test selected Signal(s) from MMC ---> PLC	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The program control “Program test” has been selected from the operator interface. The function is thus not yet activated.	
Signal status 0 or edge change 1 ---> 0	The program control “Program test” has not been selected from the operator interface.	
Related to	IS ”Activate program test” IS ”Program test active”	

V18000001.0 Interface signal	Machine function TEACH IN Signal(s) from MMC ---> PLC	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine function “TEACH IN” has been selected from the operator interface. The function is thus not yet activated.	
Signal status 0 or edge change 1 ---> 0	The machine function “TEACH IN” has not been selected from the operator interface.	
Related to	IS ”machine function TEACH IN” IS ”machine function TEACH IN active”	

V30000000.0 Interface signal	AUTOMATIC mode Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	AUTOMATIC mode is selected from the PLC program.	
Signal status 0 or edge change 1 ---> 0	AUTOMATIC mode is not selected from the PLC program.	
Signal not applicable to	signal “Mode change blocked” is present	
Related to	IS ”Active mode AUTOMATIC”	

V30000000.1 Interface signal	MDA mode Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	MDA mode is selected from the PLC program.	
Signal status 0 or edge change 1 ---> 0	MDA is not selected from the PLC program.	
Signal not applicable to	signal “Mode change blocked” is present	
Related to	IS ”Active mode MDA”	

V30000000.2 Interface signal	JOG mode Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	JOG mode is selected from the PLC program.	
Signal status 0 or edge change 1 ---> 0	JOG mode is not selected from the PLC program.	
Signal not applicable to	signal “Mode change blocked” is present	
Related to	IS ”Active mode JOG“	

V30000000.4 Interface signal	Mode change blocked Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	No change from the currently active mode (JOG, MDA or Automatic) possible.	
Signal status 0	Mode change is possible.	
Fig.		

V30000000.7 Interface signal	Reset Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The channel is to change to RESET condition. The currently active program is then in the program status “Aborted”. All running axes and spindles are decelerated to standstill along their acceleration curve without contour violation. The basic settings are set (e.g. G functions). The alarms are deleted if they are not POWER ON alarms.	
Signal status 0 or edge change 1 ---> 0	Channel status and program execution are not affected by this signal.	
Related to	IS ”Channel reset” IS ”All channels in Reset status”	
Special cases, errors, ...	An alarm that cancels the IS “Ready” ensures that the channel is no longer in Reset status. In order to be able to change the operating mode, Reset must be initiated.	

V30000001.0 Interface signal	Machine function TEACH IN Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine function TEACH IN is activated in AUTOMATIC mode.	
Signal status 0 or edge change 1 ---> 0	The machine function TEACH IN is not activated.	
Signal not applicable to	JOG mode is active.	

V30000001.2 Interface signal	Machine function REF Signal(s) to NCK (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine function REF is activated in JOG mode.	
Signal status 0 or edge change 1 ---> 0	The machine function REF is not activated.	
Signal not applicable to	JOG mode is active.	

V31000000.0 Interface signal	Active mode AUTOMATIC Signal(s) from NCK (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	AUTOMATIC mode is active.	
Signal status 0 or edge change 1 ---> 0	AUTOMATIC mode is not active.	

V31000000.1 Interface signal	Active mode MDA Signal(s) from NCK (NCK ---> PLC)	
Edge evaluation:	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	MDA mode is active.	
Signal status 0 or edge change 1 ---> 0	MDA mode is not active.	

V31000000.2 Interface signal	Active mode JOG Signal(s) from NCK (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	JOG mode is active.	
Signal status 0 or edge change 1 ---> 0	JOG mode is not active.	

V31000001.0 Interface signal	Active machine function TEACH IN Signal(s) from NCK (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine function TEACH IN is active in AUTOMATIC mode.	
Signal status 0 or edge change 1 ---> 0	The machine function TEACH IN is not active.	

V31000001.2 Interface signal	Active machine function REF Signal(s) from NCK (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine function REF is active in JOG mode.	
Signal status 0 or edge change 1 ---> 0	The machine function REF is not active.	

V32000000.4 Interface signal	Activate single block Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	In AUTOMATIC mode, the program is executed in single block mode; in MDA, anyway only one block can be entered.	
Signal status 0 or edge change 1 ---> 0	No effect	
Application example(s)	To test a new program, it can first be run in single block mode in order to be able to check the individual program steps more exactly.	
Special cases, errors, ...	- With tool radius compensation (G41,G42) selected, intermediate blocks are inserted if necessary. - With a series of G33 blocks, single block mode is only active if Dry Run Feed is selected. - In SBL1 single block mode, pure arithmetic blocks are not processed; these are only processed in SBL2. The preselection of SBL1 or SBL2 is carried out via the Program Control softkey.	

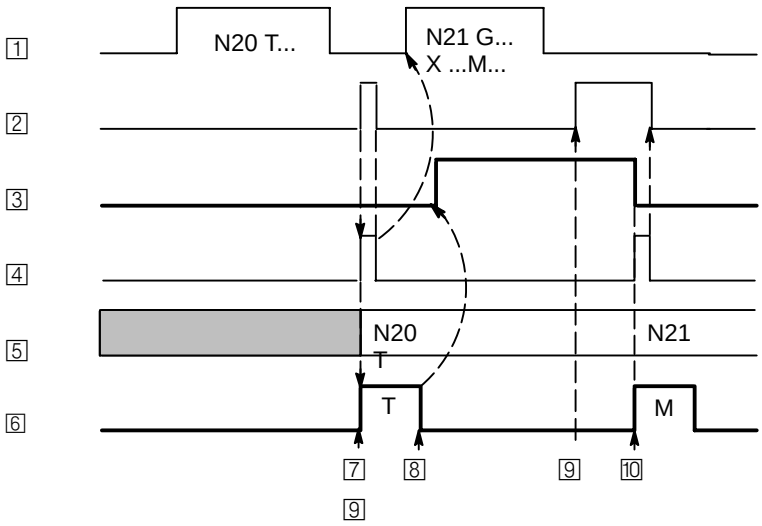
V32000000.4 Interface signal	Activate single block Signal(s) to channel (PLC ---> NCK)
Related to	IS "Single block selected" IS "Program status interrupted"
References	Section 5.2

V3200000.5 Interface signal	Activate M1 Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	When the program is run in AUTOMATIC mode or MDA, M1 contained in the part program results in programmed stop.	
Signal status 0 or edge change 1 ---> 0	M1 in the part program does not result in programmed stop.	
Related to	IS "M01 selected" (V17000000.5) IS "M0/M1 active" (V33000000.5)	

V32000001.7 Interface signal	Activate program test Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	An internal Axis Blocked signal is provided for all axes (not for the spindles). For this reason, the machine axes do not move when a part program block or part program is executed. However, the axes movements are simulated on the operator interface by changing axis position values. The axis position values for the display are generated from the calculated setpoint values. The part program is executed as usual.	
Signal status 0 or edge change 1 ---> 0	The execution of the part program is not affected by the Program Test function.	
Related to	IS "Program test selected" IS "Program test active"	

V32000002.0 Interface signal	Skip block Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Blocks in a part program which are marked with a slash (/) are skipped. If several blocks are to be skipped, this signal is only effective if it is written prior to decoding the first block, best **before "NC Start"**.	
Signal status 0 or edge change 1 ---> 0	The marked part program blocks are not skipped.	
Related to	IS "Skip block selected" IS "Program status stopped"	

V32000006.1 Interface signal	Read–in disable Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The data transfer to the interpolator is stopped for the next block. This signal is only effective in AUTOMATIC and MDA.	
Signal status 0 or edge change 1 ---> 0	The data transfer to the interpolator is enabled for the next block. This signal is only effective in AUTOMATIC and MDA.	

V32000006.1 Interface signal	Read–in disable Signal(s) to channel (PLC ---> NCK)
Application example(s)	If the auxiliary function must be completed before the next NC block is started (e.g. for tool change), automatic block change must be blocked by read–in disable.  1 Reading into buffer 2 Block executed 3 Read–in disable signal 4 Data transfer 5 Contents of interpolator 6 Output of auxiliary function 7 Data transfer to interpolator 8 Read–in disable for tool change 9 Interrogation point for read–in enable 10 Cancel read–in disable
Related to	IS ”Program status running“

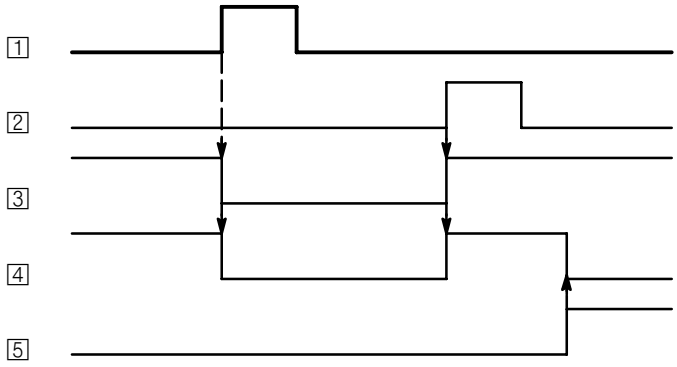
V32000006.4 Interface signal	Program level abortion Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	With each edge change 0 -> 1, the currently processed program level (subroutine level) is immediately aborted. The part program is continued from the breakpoint on the next higher program level.	
Signal status 0 or edge change 1 ---> 0	No effect	
Special cases, errors, ...	The main program level cannot be aborted with this IS but only with the IS ”Reset”.	

V32000007.0 Interface signal	NC Start inhibited Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid from SW version: 1.1
Signal status 1 or edge change 0 ---> 1	IS ”NC Start” is inactive.	
Signal status 0 or edge change 1 ---> 0	IS ”NC-Start” is active.	
Application example(s)	This signal is used, for example, to suppress a new program execution due to missing lubricant.	
Related to	IS ”NC Start”	

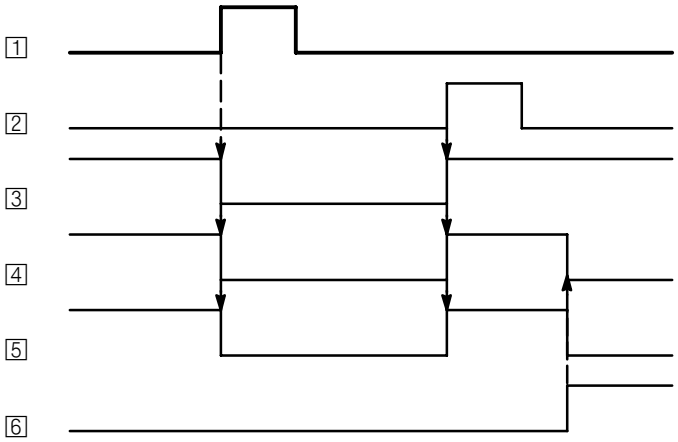
V32000007.1 Interface signal	NC Start Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	AUTOMATIC mode: The selected NC program is started or continued. If in program status ”Program interrupted”, data are transferred from the PLC to the NC, these are immediately taken into account with NC Start. MDA mode: The entered part program block is enabled for execution and continued, respectively.	
Signal status 0 or edge change 1 ---> 0	No effect	
Related to	IS ”NC Start inhibited“	

V32000007.2 Interface signal	NC Stop at block end Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The currently active NC program is stopped after the current part program block has been executed. Remaining part as with IS ”NC Stop”.	
Signal status 0 or edge change 1 ---> 0	No effect	
Related to	IS ”NC Stop” IS ”NC Stop axes plus spindles“ IS ”Program status stopped” IS ”Channel status interrupted“	

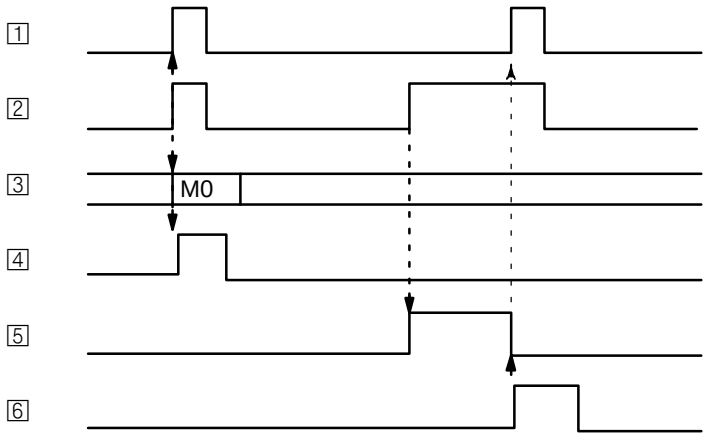
V32000007.3 Interface signal	NC Stop Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The currently active NC program is stopped immediately, and the currently active block is no longer processed. Only the axes are stopped without contour violation. Any distances to go are only traversed after restart. The program status changes to “Stopped”, and the channel status changes to “Interrupted”.	
Signal status 0 or edge change 1 ---> 0	No effect	

V32000007.3 Interface signal	NC Stop Signal(s) to channel (PLC ---> NCK)
Application example(s)	The program is continued from the breakpoint with NC Start.  1 IS "NC Stop" 2 IS "NC Start" 3 Program running 4 Axis running 5 Block executed
Special cases, errors, ...	The NC Stop signal must be provided for at least one PLC cycle time.
Related to	IS "NC Stop at block end" IS "NC Stop axes plus spindles" IS "Program status stopped" IS "Channel status interrupted"

V32000007.4 Interface signal	NC Stop axes plus spindles Signal(s) to channel (PLC ---> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The currently active NC program is stopped immediately, and the currently active block is no longer executed. Any distances to go are only traversed after restart. The axes and the spindle are stopped. They are stopped by controlled deceleration. The program status changes to Stopped, and the channel status changed to Interrupted.	
Signal status 0 or edge change 1 ---> 0	No effect	
Signal not applicable to	Channel status Reset Program status aborted	

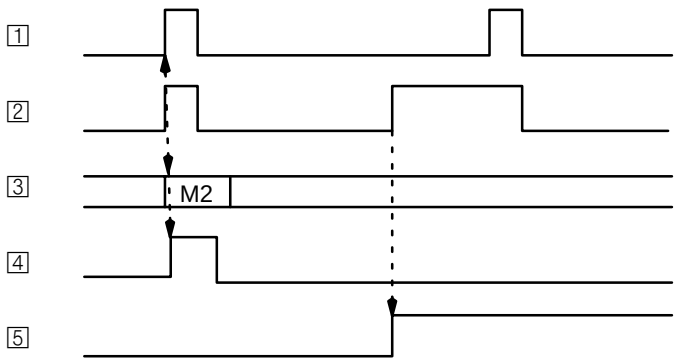
V32000007.4 Interface signal	NC Stop axes plus spindles Signal(s) to channel (PLC ---> NCK)
Special cases, errors, ...	All axes and the spindle, which have not been triggered by a program or program block (for example, axes run by means of the traversing keys on the MCP), do **not** decelerate to standstill with ”NC Stop axes plus spindles”. The program is continued from the breakpoint with NC Start. The signal ”NC Stop axes plus spindles” must be provided for at least one PLC cycle time.  1 Signal NC Stop axes 2 Signal NC Start 3 Program running 4 Axis running 5 Spindle running 6 Block executed
Related to	IS ”NC Stop at block end“ IS ”NC Stop” IS ”Program status stopped“ IS ”Channel status interrupted“

V33000000.5 Interface signal	M0/M1 active Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version
Signal status 1 or edge change 0 ---> 1	The part program block is executed, the auxiliary functions are output and - M0 is in the user memory or - M1 is in the user memory and IS ”Activate M01” is active. The program status changes to Stopped.	
Signal status 0 or edge change 1 ---> 0	- With IS ”NC Start” - With program abortion by Reset	

V33000000.5 Interface signal	M0/M1 active Signal(s) from channel (NCK ---> PLC)
Fig.	 1 Data transfer to user memory 2 Block executed 3 NC block with M0 4 M change signal (1 PLC cycle time) 5 IS ”M0/M1 active” 6 IS ”NC Start”
Related to	IS ”Activate M01“) IS ”M01 selected“

V33000001.4 Interface signal	Block search active Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The block search function is active. It has been selected and started via the operator interface.	
Signal status 0 or edge change 1 ---> 0	The block search function is not active.	
Application example(s)	The block search function can be used to jump to a certain block in the part program and start program execution only from this block.	

V33000001.5 Interface signal	M2/M30 active Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	- NC block with M2 is completely executed. If traversing movements are also programmed in this block, the signal is only output when the target position is reached.	
Signal status 0 or edge change 1 ---> 0	- No end or abortion of program - Status after turning on the control system - when starting an NC program	

V33000001.5 Interface signal	M2/M30 active Signal(s) from channel (NCK ---> PLC)
Fig.	 1 Data transfer to user memory 2 Block executed 3 NC block with M2 4 M change signal (1 PLC cycle time) 5 IS ”M2/M30 active”
Application example(s)	The PLC can detect the end of program execution by means of this signal and react on it.
Special cases, errors, ...	- The functions M2 and M30 are equivalent. Only M2 should be used. - The IS ”M2/M30 active” is statically provided after program end. - Not suitable for automatic sequential functions, such as workpiece counting, bar feed and many others. For these functions, M2 must be written in a separate block, and either the word M2 or the decoded M signal must be used. - The last block of a program may not contain auxiliary functions that lead to read–in disable.

V33000001.7 Interface signal	Program test active Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The program control “Program test” is active. The internal signal “Axes blocked” is provided for all axes (not spindles). For this reason, the machine axes do not move when a part program block or part program is executed. However, the axis movements are simulated on the operator interface by changing axis position values. The axis position values for the display are generated from the calculated setpoints. The part program is executed as normal.	
Signal status 0 or edge change 1 ---> 0	The program control “Program test” is not active.	
Related to	IS ”Activate program test” IS ”Program test selected”	

V33000003.0 Interface signal	Program status Running Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The part program has been started with IS ”NC-Start” and is running.	
Signal status 0 or edge change 1 ---> 0	- Program stopped by M00/M01 or NC Stop or mode change - The block is executed in single block mode. - End of program reached (M2) - Program aborted by Reset - Current block cannot be executed	
Special cases, errors, ...	The IS ”Program status running“ does not change to 0 when the workpiece machining is stopped by the following events: - Output of feed blocking or spindle blocking - IS ”Read–in disable“ - Feed override to 0% - Response of spindle and axis monitoring	

V33000003.1 Interface signal	Program status Waiting Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The currently active program has found a special program command in an NC block; not available with SINUMERIK 802S.	
Signal status 0 or edge change 1 ---> 0	Program status waiting not present.	

V33000003.2 Interface signal	Program status Stopped Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The NC part program has been stopped either by “NC Stop”, “NC Stop axes plus spindles”, “NC Stop at block end”, programmed M0 or M1 or single block mode.	
Signal status 0 or edge change 1 ---> 0	Program status “Stopped“ is not present.	
Related to	IS ”NC Stop” IS ”NC Stop axes plus spindles“ IS ”NC Stop at block end“	

V33000003.3 Interface signal	Program status Interrupted Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	When changing the mode from AUTOMATIC or MDA mode (with the program stopped) to JOG, the program status changes to “Interrupted”. The program can then be continued from the break-point either in AUTOMATIC mode or MDA by pressing NC Start.	
Signal status 0 or edge change 1 ---> 0	Program status “Aborted” is not present.	
Special cases, errors, ...	The IS ”Program status aborted“ indicates that the program can be continued by restart.	

V33000003.4 Interface signal	Program status Aborted Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The program is selected but not started, or the current program has been aborted by Reset.	
Signal status 0 or edge change 1 ---> 0	Program status “Aborted” is not present.	
Related to	IS ”Reset”	

V33000003.5 Interface signal	Channel status Active Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	In this channel - a part program or block is currently executed in Automatic mode or MDA; - at least one axis is traversed in JOG mode.	
Signal status 0 or edge change 1 ---> 0	”Channel status interrupted” or “Channel status Reset” is present.	

V33000003.6 Interface signal	Channel status Interrupted Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The NC part program in AUTOMATIC mode or the block in MDA mode has been interrupted either by “NC Stop”, “NC Stop axes plus spindles”, “NC Stop at block end”, programmed M0 or M1 or single block mode. The part program or the interrupted traversing movement can be continued after NC Start.	
Signal status 0 or edge change 1 ---> 0	”Channel status active” or “Channel status Reset” is present.	

V33000003.7 Interface signal	Channel status Reset Signal(s) from channel (NCK ---> PLC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The signal is set to “2” as soon as the channel is in Reset status, i.e. no processing is active.	
Signal status 0 or edge change 1 ---> 0	The signal is set to “0” as soon as soon as any processing is active in the channel, e.g. execution of a part program or block search.	

Compensation

6

Background

The accuracy of machine tools is affected by deviations from the ideal geometry, errors in the power transmission and in the measuring systems. When large workpieces are machined, temperature variations and mechanical forces often lead to a high loss in precision.

Generally, some of these deviations can be measured during the start–up of the machine and compensated during operation.

Compensations

Because of the rising demands on the accuracy of machine tools, state–of–the–art CNCs posses intelligent functions for the compensation of essential errors.

The following compensations can be activated axis–specifically:

- Backlash compensation
- LEC
  (lead error and measuring system error compensation).

The compensation functions can be set for every machine separately by means of machine data.

For a spindle with position control (positioning mode) or an axis with analog drive

- automatic drift compensation

can be activated.

6.1 Backlash Compensation

Mechanical backlash

The power transmission between a moved machine part and its drive (e.g. reversal backlash of leadscrew) is usually characterized by small backlashes, since completely backlash–free adjustment of the mechanics would result in a too high machine wear.
Furthermore, backlash can occur between the machine part and the measuring system.

Effect

If **indirect measuring systems** are used for the axes/spindles, the mechanical backlash leads to a falsification of the travel. For example, in the case of reversal of the direction, the axis will traverse by the amount of the backlash too less or too much (see Fig. 6-1 and Fig. 6-2).
The same applies to axes equipped with stepper motors (withut encoder). The encoder are considered here as “internally” existing.

Compensation

For backlash compensation, the axis–specific actual value is corrected by the backlash value with each change of the direction of the axis/spindle.

This amount can be entered for each axis/spindle in the MD: BACKLASH (reversal backlash) during start–up.

Activation

After reference–point approach, the backlash compensation is active in all operating modes.

Positive backlash

The encoder is always in advance of the machine part (e.g. table). Because the actual position acquired by the encoder is thus also in advance of the real actual position, the table will traverse too short (see Fig. 6-1). The backlash compensation value must be entered here as a **positive** value (= normal case).

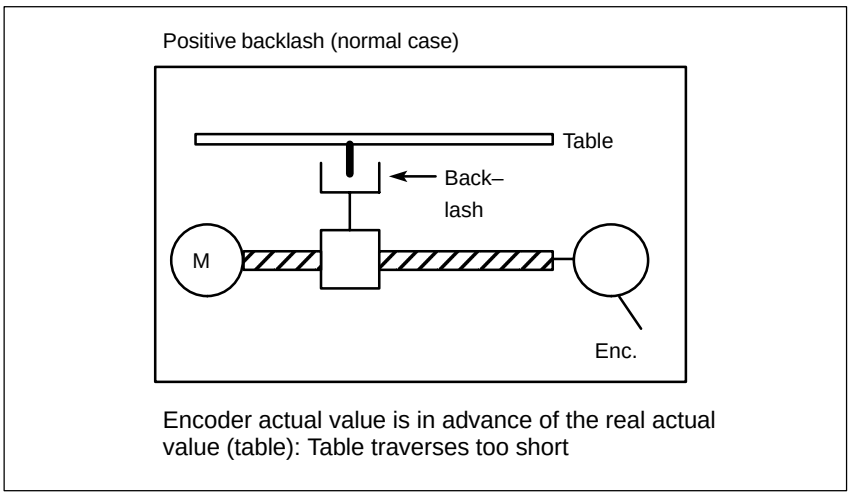


Fig. 6-1 Positive backlash (normal case)

Negative backlash

The encoder runs behind the machine part (e.g. table); the table will traverse too far (see Fig. 6-2). The compensation value must be entered as a **negative** value.

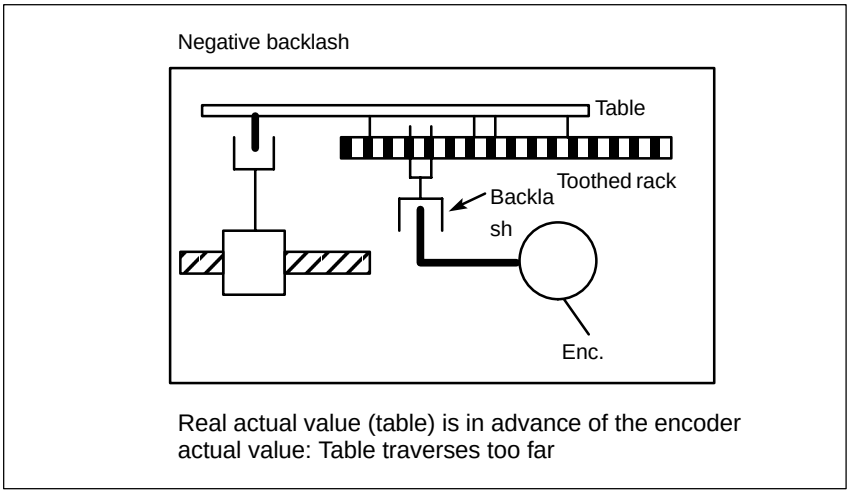


Fig. 6-2 Negative backlash

Compensation value display

The service display (operating area “Diagnosis”) displays the effective compensation value (Fig. “Service axes”, “Abs. comp. value”), in addition to the current actual position. This display value is the total of the compensation value from LEC and backlash compensation.

Large compensation values

The backlash compensation value occurred in the event of direction reversal of the axis concerned can be split over several sections. This avoids specific axis errors as caused by a too large jumps of the axis setpoint.

The contents of the axis MD36500 ENC_CHANGE_TOL are used to define the increment for the backlash compensation value (MD32450 BACKLASH).

It should be taken into account that the backlash compensation is only included in calculation after n (=MD32450 / MD36500) servo cycles. Too high a time interval may cause zerospeed monitoring alarms.

If MD: ENC_CHANGE_TOL is greater than MD BACKLASH, the compensation is carried out in a servo cycle.

6.2 Lead Error and Measuring System Error Compensation (LEC)

Function

The lead error or measuring system error compensation (in the following called “**LEC**”) is an **axial compensation.**

With the LEC, the axis–specific actual position value is modified by the associated compensation value and traversed by the machine axis immediately. A positive compensation value results in a movement of the associated machine axis in negative direction.

The amount of the compensation value is not limited and is also not monitored. In order to avoid inadmissibly high velocities and accelerations of the machine axis due to the compensation, the compensation values should be selected accordingly small. Otherwise, if large compensation values are selected, other axis monitoring functions could cause alarm messages (e.g. contour monitoring, speed rated value limiting).

Activation

The LEC is only effective if the following preconditions are fulfilled:

- The compensation values are stored in the NC user memory and effective (after Power ON).
- The function was activated for the machine axis concerned (MD:ENC_COMP_ENABLE [0] = 1).
  This is at the same time write protection for the value table.
- The axis was referenced (IS: ”Referenced/Synchronized 1“ V390x0000.4).

Once these conditions are fulfilled, the axcis–specific actual position value is modified by the associated compensation value in all operating modes and traversed by the machine axis immediately.

If the reference gets lost thenafter, for example, since the encoder frequency has been exceeded (IS ”Referenced/Synchronized 1“ = ‘0’), compensation processing is switched off.

Compensation interpolation points

The number of reserved interpolation points of the compensation table must be defined for each machine axis and for each measuring system using the MD:
MM_ENC_COMP_MAX_POINTS, and the memory required for this must be reserved.

MM_ENC_COMP_MAX_POINTS[0,AXi]

with: AX1=X axis, AX3=Z axis

Compensation table

The position–related compensation values for the axes are stored in a compensation table in the form of system variables.

The following measuring–system specific parameters must be defined for the table (see Fig. 6-3):

- **Distance between the interpolation points** ($AA_ENC_COMP_STEP[0,AXi])
  The distance between the interpolation points defines the distance between the compensation values of the associated compensation value table (meaning e and AXi see above).
- **Start position** ($AA_ENC_COMP_MIN[0,AXi])
  The start position is the axis position at which the compensation table for the axis concerned starts (≙ interpolation point 0).

  The compensation value associated to the start position is $AA_ENC_COMP[0,**0**,AXi)]

  For all positions less than the start position, the compensation value of interpolation point 0 is used (not applicable to tables with modulo).
- **Compensation value for interpolation point N of the compensation table** ($AA_ENC_COMP [e,**N**,AXi])

  The compensation value for each individual interpolation point (axis position) must be entered into the table.

  The interpolation point N is limited by the number of the maximum possible interpolation points of the associated compensation table (MM_ENC_COMP_MAX_POINTS).

  The amount of the compensation value is not limited.
  Permissible range from N: $0 \leq N <$ MM_ENC_COMP_MAX_POINTS -1
- **End position** ($AA_ENC_COMP_MAX[0,AXi])
  The end position is that axis position at which the compensation table for the axis concerned ends (≙interpolation point k).

  The compensation value associated to the end position is $AA_ENC_COMP[0,**k**,AXi)]

  For all positions greater than the end position, the compensation value of interpolation point k is used.

The number of the required interpolation points is calculated as follows:

$$k = \frac{\$AA_ENC_COMP_MAX - \$AA_ENC_COMP_MIN}{\$AA_ENC_COMP}$$

with $0 \leq k <$ MM_ENC_COMP_MAX_POINTS

The following marginal conditions shall apply to the interpolation point:

– with k = MM_ENC_COMP_MAX_POINTS - 1

⇒ The compensaiton table is used completely!

– with k < MM_ENC_COMP_MAX_POINTS - 1

⇒ The compensation table is not completely used; the compensation values in the table which are greater than k are ineffective.

- with k > MM_ENC_COMP_MAX_POINTS - 1

⇒ The compensation table is limited by the control system internally by reducing the end position; compensation values less than k are ineffective.

Caution

When entering the compensation values, it should be made sure that all interpolation points within the defined area a compensation value is associated (i.e that no gaps arise). Otherwise, for these interpolation points, the compensation value will be used which remained at these points from previously made entries.

Note

- Table parameters that contain position information are interpreted with MD: SCALING_SYSTEM_IS_METRIC=0 in inch.
- The compensation table can only be loaded if the machine data ENC_COMP_ENABLE=**0** is set. Value =1 results in activation of the compensation and thus in write protection.
- Save your compensation values by selecting the softkey “Save data“ in the operator interface Diagnosis –> Start–up (see also “Operation and Programming“)

Example

The following example shows the compensation values specified for machine axis X by means of a part program.

```
%_N_EECDAT_EEC_INI                ; Compensation table for X
$AA_ENC_COMP_STEP[0,X]   = 1.0    ; Distance between interpolation
                                    points 1.0 mm
$AA_ENC_COMP_MIN[0,X]    = –200.0 ; Compensation starts at
                                    –200.0 mm
$AA_ENC_COMP_MAX[0,X]    = 600.0  ; Compensation ends at
                                    +600.0 mm
$AA_ENC_COMP[0,0,X]      = 0.01   ; 1st compensation value
                                    (≙interpolation point 0)
                                    +0.01mm
$AA_ENC_COMP[0,1,X]      = 0.012  ; 2nd compensation value
                                    (≙interpolation point 1)
                                    +0.012mm
...                               ; etc.
$AA_ENC_COMP[0,800,X]    = –0.02  ; last compensation value
                                    (≙interpolation point 800)
                                    –0.020mm
M17                               ; End of compensation table for X
```

In this example, the number of compensation interpolation points must be MM_ENC_COMP_MAX_POINTS $\geq$ 801; otherwise, alarm 12400 is output.

The compensation table for this example requires 6.4 kBytes of the buffered NC user memory (8 bytes per compensation value).

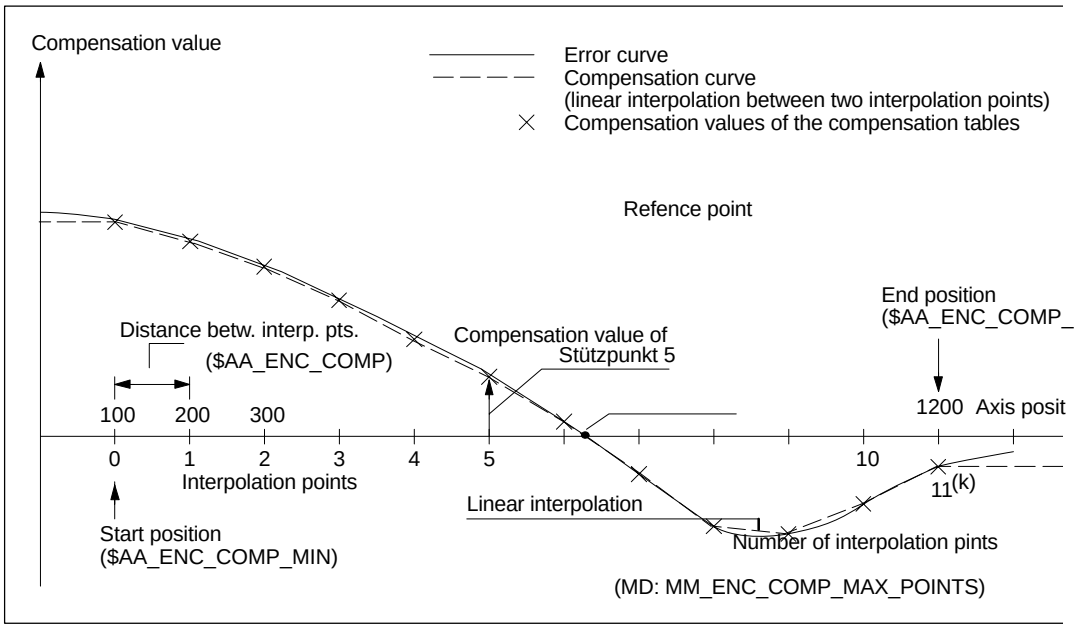


Fig. 6-3 Parameters of the compensation table (system variables for LEC)

6.3 Drift Compensation

Drift

applies only to position–controlled spindle, not for axes with stepper motors:

The temperature–depenedent drift in analog components requires that analog speed control loops must be controlled with a speed rated value unequal to zero in order to reach standstill. The position controller can only generate this speed rated value if a small following error arises on its input even on standstill. The axis/spindle will therefore leave its set position only slowly until the speed rated value that has been established due to the existing following error so large that it corresponds to the temperature drift.

Compensation

To avoid this static error, a small additional speed setpoint is provided which consists of the folowing components (see Fig. LEERER MERKER):

1. Drift basic value (MD 36720: DRIFT_VALUE)
   The value entered in MD 36720: DRIFT_VALUE is always added as an additional speed setpoint. The drift basic value is always effective. The input is always performed as a percentage with reference to the maximum cpntrolled quantity.
2. Automatic drift compensation (MD 36700: DRIFT_ENABLE)
   MD 36700: DRIFT_ENABLE = 1 (automatic drift compensation) can be used to enable automatic drift compensation for position–controlled axes/spindle.
   The control system will determine the required drift additional value at the standstill of the axes/spindle (IS ”Axis/spindle stopped” (V390x0001.4) is active) to make sure that the following error becomes 0 (adjustment crierion).

The entire drift value is the total of drift basic value and drift additional value.

The **automatic drift compensation** for a position–controlled spindle/axis is carried out under the following conditions:

- Axis/spindle is at a standstill
- No traversing request is present for the axis/spindle

DRIFT_LIMIT

The amount of the drift additional value calculated during the automatic drift compensation is internally limited with MD 36710: DRIFT_LIMIT (drift limit value with automatic drift compensation). If the drift additional value exceeds the value entered in MD: DRIFT_LIMIT, alarm 25070 “Drift value too high” is output and the drift additional value is limited to this value. The value is entered as a percentage with reference to the maximum of the controlled quantity (100 %).

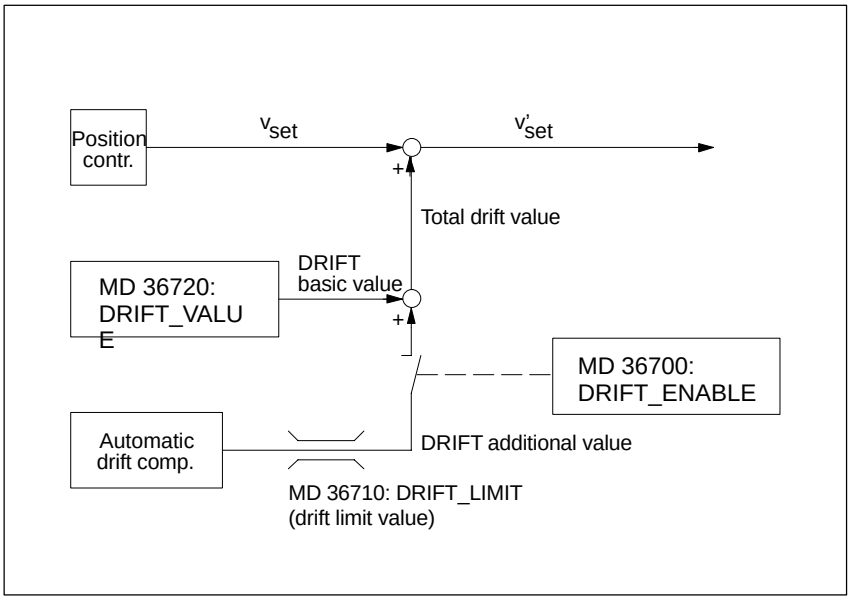


Fig. 6-4 Zusammensetzung des Drift–Drehzahlsollwertes

Service display

The effect of the drift compensation can be checked using the displayed following error in the Operating area “Diagnosis” in the menu “Service display”.
On standstill of the spindle, the displayed following error should be “0”.

Note

When direct measuring systems are used and “Automatic drift compensation” is enabled (MD: DRIFT_ENABLE=1), this results in oscillation of the axis concerned due to the mechanical backlash. Sometimes, it can be better in such cases not to work with automatic drift compensation.

6.4 Data Descriptions

Machine data

32450 MD number	BACKLASH[0] Backlash		
Default: 0	Min. input limit: ***	Max. input limit: ***	
Changes effective after NEW_CONF	Protection level: 2/7	Unit: mm or degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Backlash between positive and negative traversing direction. The input of the compensation value is • positive if the encoder is in advance of the machine part (normal case) • negative if the encoder runs after the machine part. When “0” is entered, the backlash compensation is ineffective. After reference–point approach, the backlash compensation is active in all operating modes..		
Special cases, errors, ...			

32700 MD number	ENC_COMP_ENABLE[0] LEC active [n]		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	1: The LEC is activated for the axis/measuring system. The LEC can be used to compensate lead screw errors and measuring system errors. The function is internally only enabled if the respective machine data is referenced (IS: “Referenced/Synchronized = 1). Write protection function (compensation values) active. 0: The LEC for the axis is not active.		
Related to	MD: MM_ENC_COMP_MAX_POINTS number of interpolation points with LEC IS “Referenced/Synchronized 1 ”		

36500 MD number	ENC_CHANGE_TOL Portion of backlash		
Default: 0.1	Min. input limit: 0	Max. input limit: plus	
Changes effective after NEW_CONF	Protection level: 2/7	Unit: mm or degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Portion when provinding backlash value		
Related to	MD: BACKLASH[0] backlash compensation		

36700 MD number	DRIFT_ENABLE Automatic drift compensation		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after NEW_CONF	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	(only with position–controlled spindle or axis with analog drive) MD: DRIFT_ENABLE activates automatic drift compensation. 1: Automatic drift compensation is active With automatic drift compensation, the control system permanently calculates the additional drift value automatically which is required for the following error to come to zero (compensation criterion). 0: Automatic drift compensation is not active.		
MD not applicable to	for non–position controlled spindles		

36710 MD number	DRIFT_LIMIT Drift limit value for automatic drift compensation		
Default: 0	Min. input limit: 0	Max. input limit: plus	
Change valid after NEW_CONF	Protection level: 2/7	Unit: % of controlled quantity (e.g. 10 V ≙ 100%)	
Data type: DOUBLE	Valid as from SW version: 3		
Meaning:	This MD can be used to limit the amount of the drift additional value determined during the automatic drift compensation. If the drift additional value exceeds the limit value entered in MD: DRIFT_LIMIT, alarm 25070 “Drift value too large” is displayed and the drift additional value is limited to this value.		
MD not applicable to	MD: DRIFT_ENABLE = 0		
Related to	MD: DRIFT_ENABLE (automatic drift compensation)		

36720 MD number	DRIFT_VALUE Drift basic value		
Default: 0	Min. input limit: 0	Max. input limit:	
Change valid after NEW_CONF	Protection level: 2/7	Unit: %	
Data type: DOUBLE	Valid as from SW version: 3		
Meaning:	The drift basic value entered in MD: DRIFT_VALUE is always delivered as an additional speed value. The drift basic value is **always** effective (independently of MD: DRIFT_ENABLE). Whereas the automatic drift compensation is only effective for analog position–controlled axes/ spindles, the drift basic value is also effective for a speed–controlled spindle.		
MD not applicable to			

38000 MD number	MM_ENC_COMP_MAX_POINTS[0] Number of interpolation points with LEC (SRAM)		
Default: 0	Min. input limit: 0	Max. input limit: 5000	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: DWORD	Valid as from SW version:		
Meaning:	The number of interpolation points required for the LEC must be defined. The required number can be calculated as follows using the defined parameters: $$MD: MM_ENC_COMP_MAX_POINTS = \frac{\$AA_ENC_COMP_MAX - \$AA_ENC_COMP_MIN}{\$AA_ENC_COMP_STEP} + 1$$ $AA_ENC_COMP_MIN start position (system variable) $AA_ENC_COMP_MAX end position (system variable) $AA_ENC_COMP_STEP dist. betw. interp. pts. (system variable) When selecting the number of interpolation points or the distance between them, the size of the compensation table and the required memory capacity of the buffered NC user memory (SRAM), which results from this should be taken into account. 8 bytes are needed per compensation value (interpolation point).		
Special cases, errors, ...	**Caution:** After the MD: MM_ENC_COMP_MAX_POINTS has been modified, the buffered NC user memory is set up automatically during power up of the system. During this process, all data of the buffered NC user memory (e.g. part programs, tool offests, etc.) get lost. The alarm 6020 ”Machinen data modified - memory mapping modified“ is signaled. If the NC user memory cannot be mapped, since the available total memory is not sufficient, alarm 6000 ”Memory mapping carried out with standard machine data” is signaled. In this case, as an alternative, the NC user memory mapping is carried out with the default values of the standard machine data.		
Related to	MD: ENC_COMP_ENABLE[0] SSFK active		

Face Axis

7

Brief description

With **control systems for turning machines**, the X axis is defined as face axis. For these reason, some special functions are bound to this axis.

- Radius/ diameter specification –G22/23
- The axis provides the actual path values for the function “Constant cutting speed” – G96

7.1 Radius / Diameter Programming: G22, G23

Functionality

To machine part on turning machines, it is common practice to program the position information for the X axis (face axis) as diameter dimension. The control system will interpret this value as diameter only for this axis.
If necessary it is possible to change to radius programming in the program.

Programming

```
G22     ;Radius input
G23     ;Diameter input
```

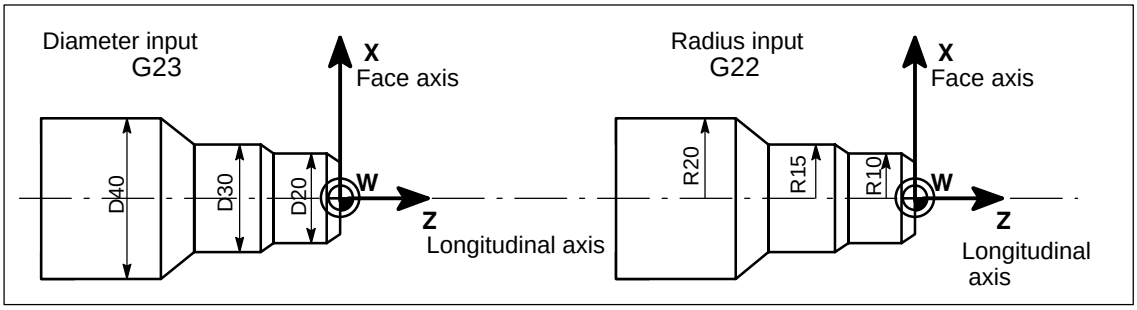


Fig. 7-1 Diameter and radius input for the face axis

Information

G22 or G23 evaluates the end point specification for the X axis as radius or diameter specification.

A programmable offset with G158 X... is always interpreted as radius specification.

Programming example

```
N10 G23 X44 Z30          ;for X axis  diameter
N20 X48 Z25              ;G23 is still active
N30 Z10
...
N110 G22 X22 Z30         ;Change to radius input for
                          X axis from here
N120 X24 Z25
N130 Z10
...
```

Setpoint/actual value display

If the function G23 is active for the face axis, the position values in the workpiece coordinate system (WCS) are displayed as a diameter value.The display in the machine coordinate system (MCS) is always carried out with radius dimension.

7.2 Constant Cutting Speed: G96

Functionality

Precondition: This function requires a controlled spindle.
With the function G96 switched on, the spindle speed is adapted to the diameter of the currently machined workpiece (face axis) such that the programmed cutting speed S at the tool edge remains constant (spindle speed multiplied with diameter = constant).
From the block with G96, the S word is interpreted as cutting speed. G96 is modal and is active until it is canceled by another G function of the group (G94, G95, G97).

Programming

```
G96 S... LIMS=... F...          ;Constant cutting speed ON
G97                             ;Constant cutting speed OFF
```

AWL	
S	Cutting speed, unit m/min
LIMS=	Upper limit speed of spindle, only effective with G96
F	Feed specified with the unit mm/rev – as with G95

Note: The feed F is interpreted here always in the unit mm/rev.
If previously G94 was active instead of G95, a new appropriate F value must be entered!

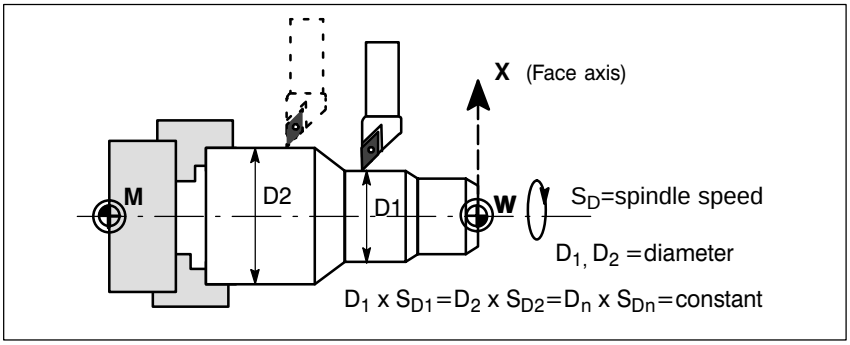


Fig. 7-2 Constant cutting speed G96

Information

For more detail information refer to the User Manul “Operation and Programming”.

Reference–Point Approach

8

8.1 Fundamentals

Why referencing?

To ensure that the control system finds machine zero exactly after it has been switched on, the control must be synchronized with the measuring systems connected to the axis or spindle. In axes context, this process is called 'referencing'. It is also required for axes with stepper motors that do not have a position measuring system. In this context, the position measuring system is considered "internally existing".

Operation of reference–point approach

Approach to the reference point can be started for each machine axis in the operating mode JOG / Reference–Point Approach by means of a direction key, depending on the MD: REFP_CAM_DIR_MINUS. All further actions are carried out automatically. When an axis is referenced, it is displayed on the screen (see User Manual "Operation and Programming"). It is possible to reference all axes at the same time.

If the machine axes are to be referenced in a certain order, either the operator must observe the specified order when starting or channel–specific referencing is set.

Synchronization signal and reference cams

The synchronization signal is usually provided by the **zero pulse** of an incremental encoder. If no measuring system exists (axis with stepper motor without position measuring system), a **BERO** sensor (ranging sensor) is required. This can be installed directly on the motor shaft or spindle. In this case, the pulses are provided during each revolution. Another signal must be provided in order to differentiate which pulse is used to synchronize the axis. This signal is provided from a **reference cam**. The signals of the reference cam are also used to control the automatic sequence of reference–point approach.
Synchronization can be performed with the rising edge of the BERO sensor (single–edge evaluation) or with the BERO–edge middle (double–edge evaluation).

If only one synchronization signal is provided over the entire traversing range, no referencing cam is required (MD: REFP_CAM_IS_ACTIVE =0).

To transfer the switching edge signal of the BERO to the control system, a high–speed input is required. To this aim, the SINUMERIK 802S provides the inputs on connector X20 (connector for high–speed inputs) and the inputs on pin 13 (for the X axis), pin 14 (for the X axis) and pin 15 (for the Z axis) (see Technical Manual "Start–Up").
The cam signal is acquired via a PLC input and transferred to the NC as an interface signal (IS "Reference–point approach delay" V 380x1000.7).

Interface signals

When the machine function REF in JOG mode (IS “Machine function REF”) (V30000001.2) is active, reference–point approach is possible. As a feedback information, the IS “Active machine function REF” (V31000001.2) must be present.
Axis–specific reference–point approach is started for each machine axis separately with the IS “Traversing keys plus/minus” (V380x0004.6 and .7).

Special characteristics

- The IS ”Reset” (V30000000.7) aborts reference–point approach. All axes not yet reached their reference points are considered not referenced. A respective alarm is displayed.
- The monitoring function “ Software limit switches“ is only effective for referenced machine axes.
- During reference–point approach, the axis–specific accelerations are always within the specified values (exception: in case of alarms).
- When referencing is started, the direction key is only active for the direction stored in MD: REFP_CAM_DIR_IS_MINUS.
- Whether NC programs in the modes AUTOMATC or MDA can be started, depends on the MD: REFP_NC_START_LOCK. If “1” is set, the program can only be started if **all** axes to be referenced have been referenced.
- The interface signal IS ”Referenced/synchronized 1” (V390x0000.4) displays whether an axis has been referenced.

8.2 Referencing Axes

Time sequence

The referencing sequence of axes with reference cams and of stepper motor axes without measuring system can be divided into three phases:

- Phase 1: Reference–point approach
- Phase 2: Synchronization with synchronous pulse
  (BERO signal with stepper motor axes or zero pulse of an incremental measuring system with axes equipped with an analog drive)
- Phase 3: Reference–point approach

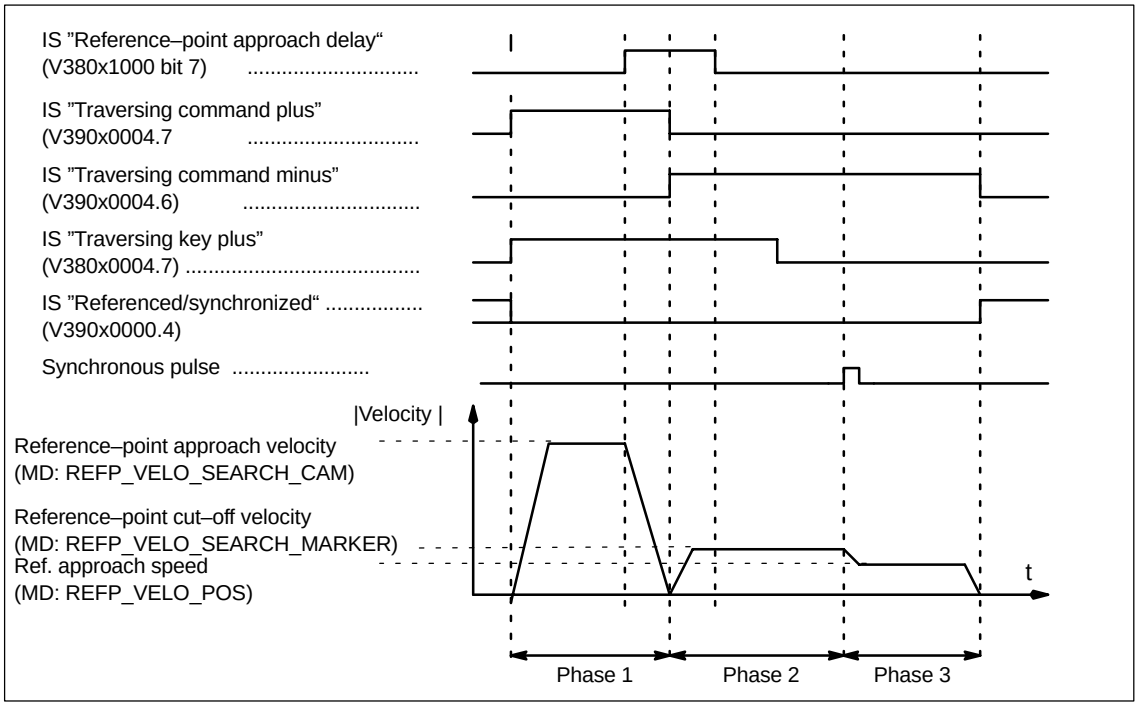


Fig. 8-1 Example of a sequence of interface signals (IS), BERO single–edge evaluation

Properties

When traversing to the reference point cam (phase 1)

- Feed Override and Feed Stop are effective.
- The machine axis can be stopped/started with NC Stop/NC Start.
- If the machine axis does not stop on the reference cam (e.g. cam too short), an appropriate alarm is output.

When synchronizing with the synhronous pulse (phase 2)

- The feed override is not effective; a feed override of 100 % applies. If the feed override is 0 %, the movement is canceled.
- Feed Stop is effective; the axis will stop and an appropriate alarm is displayed.
- The machine axis cannot be stopped/started with NC Stop/NC Start.

When traversing to the reference point (phase 3)

- Feed Override and Feed Stop are effective.
- The machine axis can be stopped/started with NC Stop/NC Start.
- If the reference point offset is less than the braking path of the machine axis from approach speedto the standstill, the reference point is approached from the opposite direction.

Reference cam

What must be the minimum length of the reference cam?
The length of the reference cam must such that the deceleration process is stopped on the cam (standstill on the cam) when the cam is approached and the cam is left with reference–point cut–off velocity in the opposite direction (leaving with constant velocity).

To calculate the minimum length of the cam, the higher of the following speeds must be entered in the formula below:

$$\text{Minimum length} = \frac{(\text{Reference–point approach speed or -cut–off speed})^2}{2 \cdot \text{axis acceleration}}$$

If the machine axis cannot stop on the reference cam (IS ”Reference–point approach delay” (V380x1000.7 is reset), alarm 20001 is output. Alarm 20001 can only occur if the reference cam is too short and the machine axis moves across the reference point during deceleration in phase 1.

If the reference cam reaches up to the traversing end of the axis, an inadmissible start point for referencing (behind the cam) is ruled out.

Reference cam adjustment

The reference cam must be adjusted exactly.

The time response that determines reference–point detection by the control (NCK) depends on the following factors:

- Switching accuracy of the reference point cam
- Reference–point switch delay (normally closed contact)
- Delay on PLC input
- PLC cycle time
- Internal processing time

The practice has shown that the adjustment of the reference–point edge required for synchronization in the middle between two BERO signals (or zero pulses) is the best solution.

Warning

If the reference cam is not adjusted exactly, a false synchronous pulse (BERO, zero mark) could be evaluated. As a consequence, the control system will assume a wrong machine zero and traverse the axes to the wrong positions; the software limit switches will also be enabled for the wrong positions and can thus not protect the machine.

Referencing without reference–point cam

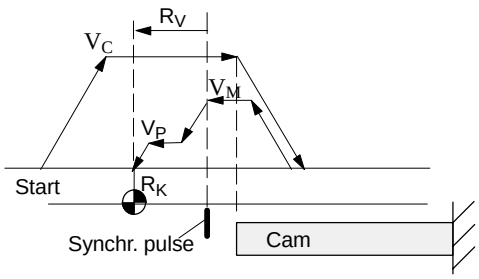
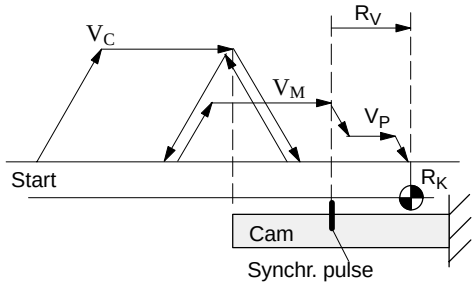
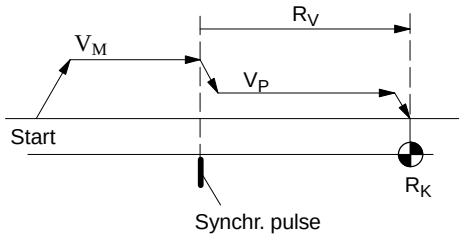
A machine axis does not require a reference–point cam if it provides only one synchronous pulse over the entire traversing range.

If axes are referenced without cam, the synchronization is carried out as follows (only phases 2 and 3):

- Synchronization with pulse
- Approach to reference point

Sequence of motions

The following Table shows the individual sequence of motions for referencing with/without reference–point cam.

Referencing Method	Synchr. Pulse	Sequence of Motions
with reference–point cam	Synchronous pulse ahead of cam, reference–point coordinate ahead of synchr. pulse	
	Synchronous pulse on cam, reference–point coordinate after synchr. pulse, on cam = with reversal	
without reference cam	Reference coordinate after synchr. pulse	

V_C- Reference–point approach velocity (MD: REFP_VELO_SEARCH_CAM)
V_M-Reference–point cut–off speed (MD: REFP_VELO_SEARCH_MARKER)
V_P- Reference–point approach velocity (MD: REFP_VELO_POS)
R_V - Reference–point offset
(MD: REFP_MOVE_DIST + REFP_MOVE_DIST_CORR)
R_K - Reference–point coordinate (MD: REFP_SET_POS[0])
with reversal - MD: REFP_SEARCH_MARKER_REVERS =1

BERO signal

For stepper motor axes only:
The respective actual value is stored when the selected edge of the BERO signals is received.

In order to achieve a good repeatability of the reference point, the search speed for the BERO edge may not exceed a certain maximum value, which depends on the BERO type.

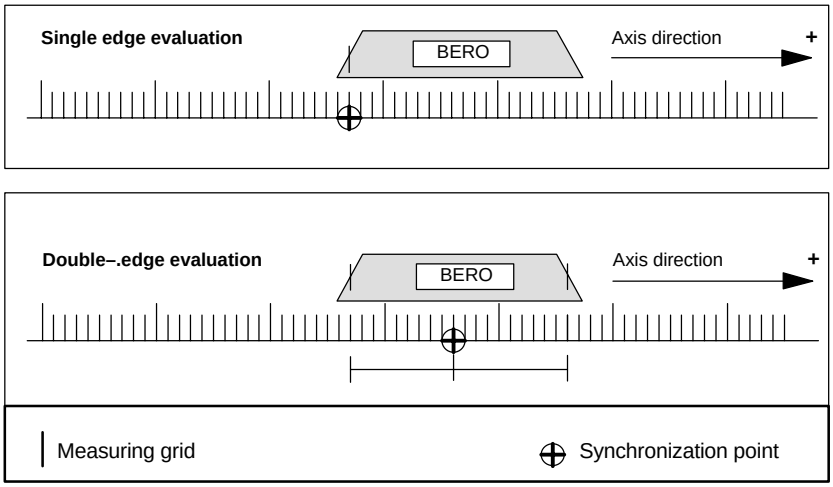


Fig. 8-2 Synchronization point definition

Single–edge evaluation

The positive edge of the BERO signal is interpreted as a synchronization mark. The associated actual value is the synchronization point.
Edge selection is carried out via MD: ENC_REFP_MODE = 2.

Double–edge evaluation

The positive and negative edges of the reference–point BERO are acrossed one after the other and the respective actual values are recorded. The average value is the synchronization point at which phase 2 ends and phase 3 starts.

The selection is carried out via MD: ENC_REFP_MODE = 4.

Due to different delay times of the two BERO edges, the synchronization point will not be exactly in the middle.

Using the same reference–point cut–off speed, the single–edge evaluation will result in better repeatabilities.

8.3 Data Descriptions

Machine data

20700 MD number	REFP_NC_START_LOCK NC Start inhibited without reference point		
Default: 1	Min. input limit: 0	Max. input limit: 1	
Changes effective after Reset	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	0: The IS "NC Start (V32000007.1) for starting part programs or part program blocks in AUTOMATIC or MDA is active even if one or all axes of the channel are **not yet referenced.** In order to ensure that the axes nevertheless reach the correct position after NC Start, the workpiece coordinate system (WCS) must always be adapted to the currently active machine coordinate system (settable zero offset determination). 1: NC Start only if all axes are referenced.		

30240 MD number	ENC_TYPE		
Default: 0	Min. input limit: 0	Max. input limit: 5	
Changes effective after POWER ON	Protection level: 2/7	Unit: -	
Data type: BYTE	Valid as from SW version:		
Meaning:	Encoder type: 0: Simulation 1: Used 2: Square–wave encoder (standard, line number multiplied with four) 3: Encoder for stepper motor (BERO) 4: Used 5: Used		
Related to			

34000 MD number	REFP_CAM_IS_ACTIVE Axis with reference–point cam		
Default: 1	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	Machine axes that have only one synchronous mark (zero pulse, BERO) over their entire traversing range can be marked as machine axes **without reference–point cam** by means of the MD: REF_CAM_IS_ACTIVE. The machine axis marked in this way accelerates to the speed set in MD: REFP_VELO_SEARCH_MARKER (reference–point cut–off speed) after the traversing key plus/minus has been pressed and synchronizes with the zero mark. During this process, make sure that the start point is always ahead of the synchronous mark.		

34010 MD number	REFP_CAM_DIR_IS_MINUS Reference–point approach in minus direction		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	**REF_CAM_DIR_IS_MINUS = 0** : Reference–point approach in plus direction (traversing key + active) **REF_CAM_DIR_IS_MINUS = 1** : Reference–point approach in minus direction (traversing key - active) --- If the machine axis has stopped **ahead of the reference cam**, it accelerates to the speed and direction specified in MD: REFP_VELO_SEARCH_CAM (reference–point approach speed), irrespective of whether the traversing key plus/minus is pressed. If the wrong traversing key is pressed, reference–point approach will not be started. If the machine axis stands **on the reference cam,** it accelerates to the speed specified in MD: REFP_VELO_SEARCH_CAM (reference–point approach speed) and traverses in the direction opposite to the direction specified in MD: REFP_CAM_DIR_IS_MINUS. A machine axis (start point) **behind the reference cam** must be ruled out.		

34020 MD number	REFP_VELO_SEARCH_CAM Reference–point approach speed		
Default: 5000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: mm/min,	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The reference–point approach speed is the speed with which the machine axis traverses in the direction of the reference cam (phase 1) after the traversing key has been pressed. The maximum amount of this value must be set such that the axis can be decelerated to standstill before it reaches a hardware switch and comes to standstill **on the reference cam.**		

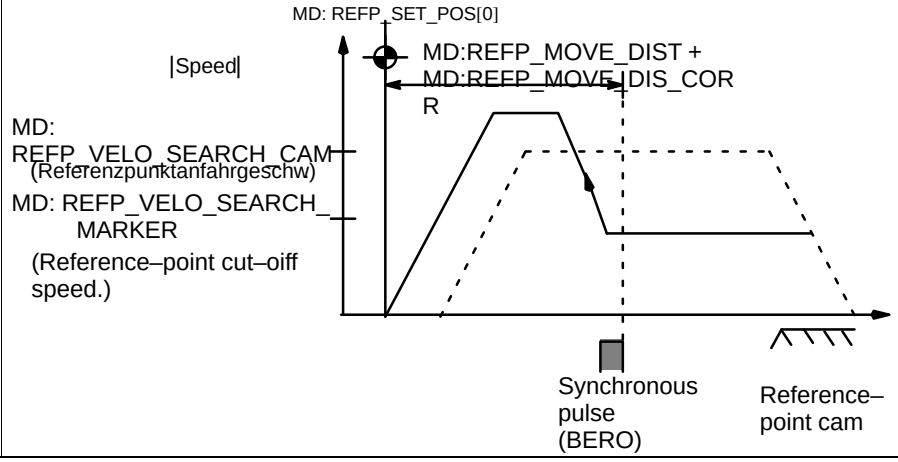
34030 MD number	REFP_MAX_CAM_DIST Max. distance to reference cam		
Default: 10000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: mm	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	If the machine axis traverses a path set in MD: REFP_MAX_CAM_DIST from the start positon in the direction of the reference cam without reaching the reference cam (IS “Reference–point approach delay” (380x1000/7) is reset), the axis stops and alarm 20000 ”Reference cam not reached“ is output.		

34040 MD number	REFP_VELO_SEARCH_MARKER[0] Reference–point cut–off speed		
Default: 300	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The axis traverses with this speed from the moment when the reference–point cam is detected to the synchronization with the first synchronization pulse (BERO, zero mark) (->phase 2). If MD: REFP_SEARCH_MARKER_REVERSE=0 (no reversal of direction due to falling reference–cam edge): The search is carried out immediately at this speed; traversing direction: always in the opposite direction to the direction set for cam search (MD: REFP_CAM_DIR_IS_MINUS). This direction remains unaffected by the new reference–cam edge. ---------- If MD: REFP_SEARCH_MARKER_REVERSE=1 (reversal of direction by falling reference–cam edge) : This speed will only be active if a rising edge of the reference cam has been detected again, i.e. traversing starts only in the direction opposite to the direction set for the cam search (MD: REFP_CAM_DIR_IS_MINUS). The speed is defined by MD: REFP_VELO_SEARCH_CAM. A falling reference–cam edge will stop the axis, turn the direction of rotation, and the search for the reference–point cut–off speed will be carried out with the first synchronous pulse.		
Related to	MD: REFP_SEARCH_MARKER_REVERSE MD: REFP_CAM_DIR_IS_MINUS		

34050 MD number	REFP_SEARCH_MARKER_REVERSE[0] Reversal of direction on reference cam		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	This machine data can be used to set the search direction of the first synchronous pulse (BERO, zero mark) (synchr. pulse ahead of or on the reference cam). **0:** Synchronization after falling reference–cam edge The machine axis accelerates to the speed set in MD: REFP_VELO_SEARCH_MARKER (reference–point cut–off speed) opposite to the direction specified in MD: REFP_CAM_DIR_IS_MINUS (reference–point approach in minus direction). When the reference point is left (IS ”Reference–point approach delay“ (V380x1000.7) is reset), the control system will synchronize itself with the first synchronous pulse (BERO, zero mark). **1:** Synchronization after rising reference–cam edge The machine axis accelerates to the speed set in MD: REFP_VELO_SEARCH_CAM (reference–point approach speed) opposite to the direction specified in MD: REFP_CAM_DIR_IS_MINUS. When the reference point is left (falling edge, IS ”Reference–point approach delay“ is reset), the machine axis decelerates to standstill and traverses to the reference cam in the opposite direction with the speed specified in MD: REFP_VELO_SEARCH_MARKER (reference–point cut–off speed). When the reference point is reached (IS “Reference–point approach delay” (380x1000.7) is set), the control system synchronizes itself with the first synchronous pulse (BERO, zero mark).		

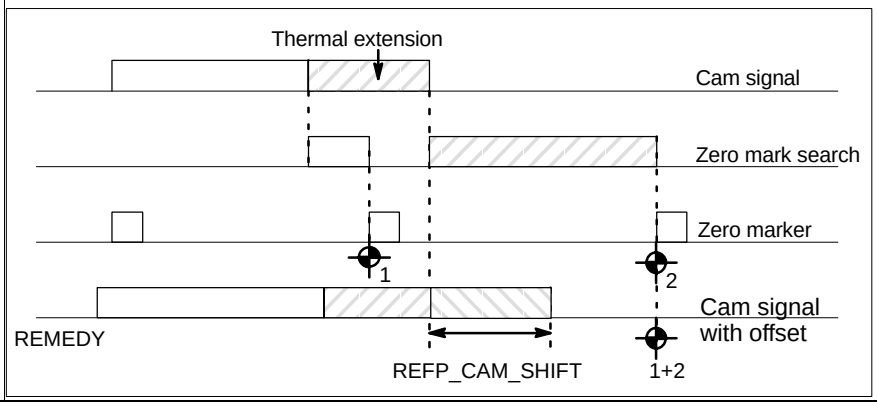
34060 MD number	REFP_MAX_MARKER_DIST[0] Max. distance to reference mark		
Default: 20	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level:2/7	Unit: mm	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Monitoring function: If the machine axis starts traversing a distance specified in MD: REFP_MAX_MARKER_DIST from the reference cam (IS "Reference–point approach delay" is reset) without detecting the reference cam, the axis stops and alarm 2002 is output.		
Application example(s)	If you wish to make absolutely sure that the control system detects that always the same synchronous pulse is used for synchronization (otherwise, a wrong machine zero is detected), the max. value in MD: REFP_MAX_MARKER_DIST may not exceed the distance between two reference marks (synchronous pulses).		

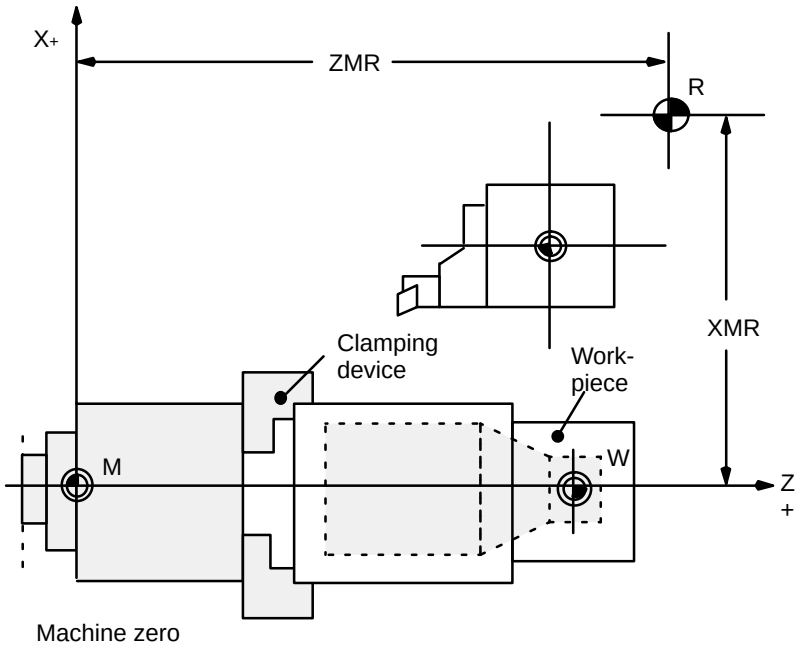
34070 MD number	REFP_VELO_POS Reference–point positioning speed		
Default: 10000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The axis traverses with this speed from the moment when the synchronous pulse is received to the moment when the reference point is reached (reference–point coordinate MD: REFP_SET_POS).		

34080 MD number	REFP_MOVE_DIST[0] Partial distance synchronous pulse <->reference point		
Default: -2.0	Min. input limit: ***	Max. input limit: ***	
Changes effective after Power On	Protection level: 2/7	Unit: mm	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	After synchronization with the synchronous pulse, the machine axis accelerates to the speed specified in MD: REFP_VELO_POS (reference point positioning speed) and traverses a distance which results from adding the distances specified in MD: REFP_MOVE_DIST und MD: REFP_MOVE_DIST_CORR (reference point offset) (-> phase 3) This distance determined by adding is exactly the distance between the detected synchronous pulse and the reference point.  		

34090 MD number	REFP_MOVE_DIST_CORR[0] Reference–point offset		
Default: 0	Min. input limit: ***	Max. input limit: ***	
Changes effective after Power On	Protection level: 2/7	Unit: mm, degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	After the synchronous mark has been detected, the axis is positioned away from the synchronous mark by the distance REFP_MOVE_DIST + REFP_MOVE_DIST_CORR. When this distance is traversed, the axis has reached its reference point. REFP_SET_POS is transferred to the actual value. During the traversing movement by REFP_MOVE_DIST+REFP_MOVE_DIST_CORR, the override switches are effective.		

34092 MD number	REFP_CAM_SHIFT Electronic reference–cam offset for incremental measuring systems with equidistant zero marks	
Default : 0.0	Min. input limit: 0.0	Max. input limit:
Changes effective after Power On	Protection level: 2/7	Unit: mm
Data type: DOUBLE	Valid as from SW version:	
Meaning:	When the reference–cam signal is received, zero mark search is started not immediately but with a delay corresponding to the distance REFP_CAM_SHIFT. The repeatability of zero mark search can thus be ensured by defined zero mark selection even in case of temperature–dependent extension of the reference cam. Since the reference cam offset is calculated by the control system in the interpolation cycle, the real cam offset is at least REFP_CAM_SHIFT and max. REFP_CAM_SHIFT+(REFP_VELO_SEARCH_MARKER/interpolation cycle). The reference–cam offset acts in the search direction of the zero mark. The reference–cam offset is only active if the cam REFP_CAM_IS_ACTIVE=1 is present.	



34100 MD number	REFP_SET_POS[0] Reference point (reference–point coordinate)		
Default: 0.0	Min. input limit: ***	Max. input limit: ***	
Changes effective after RESET	Protection level: 2/7	Unit: mm, degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The position value that is **set** as the current **axis position** after the synchronous mark has been detected and the distance REFP_MOVE_DIST + REFP_MOVE_DIST_CORR has been traversed.		
Fig. 12	 M Machine zero W Workpiece zero R Reference point XMR Reference–point value in X direction (MD: REFP SET POS [X]) ZMR Reference–point value in Z direction (MD: REFP SET POS [Z])		
Related to			

34110 MD number	REFP_CYCLE_NR Axis order with channel–specific referencing		
Default: 0	Min. input limit: -1	Max. input limit: 5	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BYTE	Valid as from SW version:		
Meaning:	**0:** -----> axis–specific referencing, no channel–specific referencing for this axis Axis–specific referencing is started for each machine axis separately by means of the IS ”Traversing keys plus/minus” (V380x0004). If the machine data axes to be referenced in a certain order, the operator must provide for the desired order prior to the start. **>0:** channel–specific referencing Channel–specific referencing is started by means of the IS ”Activate referencing“ (V32000001.0). The control system acknowledges the successful start by the IS ”Referencing active“ (V33000001.0). Channel–specific referencing can be used to reference each machine axis assigned to the channel (to this aim, the traversing keys plus/minus are simulated internally in the control system). The axis–specific MD: REFP_CYCLE_NR can be used to define the machine axis referencing order. **1:** The machine axis is started by channel–specific referencing first. **2:** The machine axis is started by channel–specific referencing if all machine axes that are marked in MD: REFP_CYCLE_NR with “1” are referenced. **3:** The machine axis is started by channel–specific referencing if all machine axes that are marked in MD: REFP_CYCLE_NR with “2” are referenced. **4:** Analogous for further machine axes. **–1:** The machine axis is not started by channel–specific referencing and NC Start is possible without referencing this axis. Note: The effect of an entry of -1 for all axes of a channel can be achieved by setting a channel–specific MD: REF_NC_START_LOCK (NC Start inhibited without referencing) to zero).		
MD not applicable	to axis–specific referencing		
Related to	IS ”Activate referencing” (V32000001.0) IS ”Referencing active“ (V33000001.0)		

34200 MD number	ENC_REFP_MODE[0] Position measuring system type	
Default: 1	Min. input limit: 1	Max. input limit: 6
Changes effective after Power On	Protection level: 2/7	Unit: -
Data type: BYTE	Valid as from SW version:	
Meaning:	For referencing, the position measuring systems installed can be divided into two groups using MD: ENC_REFP_MODE (position measuring system type): **0:** No reference–point approach possible **1:** (Spindle or axis with analog drive) Referencing with incremental measuring systems, zero pulse on encoder track: incremental rotatary measuring system **2:** BERO with 1-edge detection for stepper motor **3:** (not available) **4:** Bero with 2-edge evaluation for stepper motor	

36310 MD number	ENC_ZERO_MONITORING Zero mark monitoring		
Default: 0	Min. input limit: 0	Max. input limit: ***	
Change valid after NEW_CONF	Protection level: 2/7	Unit: -	
Data type: DWORD	Valid as from SW version:		
Meaning:	This MD enables zero mark monitoring. **0:** No zero mark monitoring **>0:** Number of detected zero mark errors at which the monitoring function is to respond (alarm output) **=100:** In addition, encoder monitoring is disabled (alarms 25000, 25001)		
Related to ...			

Interface signals

Signal to/from channel

32000001.0 Interface signal	Activate referencing Signal(s) to channel (PLC -> NCK)		
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ---> 1	Channel–specific referencing is started with the IS “Activate referencing”. The control system acknowledges the successful start by the IS “Referencing active”. Channel–specific referencing can be used to reference each machine axis assigned to the channel (to this aim, the traversing keys plus/minus are simulated internally in the control system). The axis–specific MD: REFP_CYCLE_NR (axis order for channel–specific referencing) can be used to define the machine axis referencing order. When all axes entered in MD: REFP_CYCLE_NR have reached their reference point, the IS ”All axes referenced” (V33000004.2) is set.		
Application example(s)	The following possibilities are provided to reference the machine axes in a certain order: ● The operator must provide for the desired order before starting the movement. ● The PLC must check or define the order by itself during start. ● The function “Channel–specific referencing” is used.		
Related to	IS ”Referencing active” IS ”All axes to be referenced have been referenced”		

33000001.0 Interface signal	Referencing active Signal(s) to channel (NCK -> PLC)		
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ---> 1	Channel–specific referencing has been started by means of the IS “Referencing active”, and the successful start has been acknowledged by the IS “Referencing active”. Channel–specific referencing is running.		
Signal status 0 or edge change 1 ---> 0	● Channel–specific referencing is completed. ● Axis–specific referencing is running ● No referencing active.		
Signal not applicable to	spindles		
Related to	IS ”Activate referencing“		

V33000004.2 Interface signal	All axes to be referenced are referenced Signal(s) from channel (PLC -> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	All axes of the channel which must be referenced are referenced. MD: REFP_NC_START_LOCK (NC Start inhibited without reference point) is zero. If two position measuring systems are connected to an axis, what would prevent the axis from being started, the active measuring system must be activated in order to ensure that the axis is considered referenced. Only if this signal is present, NC Start for part program execution is accepted. Axes to be referenced are all axes that in parking position (position measuring systems inactive and servo enable canceled).	
Signal status 0 or edge change 1 ---> 0	One or several axes of the channel which must be referenced are not referenced.	
Special cases, errors, ...	The spindles of the channel have no effect on this IS.	
Related to	IS ”Referenced/synchronized 1” IS ”Referenced/synchronized 2”	

Signals to axis/spindle

V380x1000.7 Interface signal	Reference–point apporoach delay Signal(s) to axis/spindle (PLC -> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine axis is on the reference cam.	
Signal status 0 or edge change 1 ---> 0	The machine axis is ahead of the reference cam. It is recommended to use an accordingly long reference cam (to the traversing end) in order to prevent the machine axis from running across the reference cam.	
Related to		

Signals from axis/spindle

V390x0000.4 Interface signal	Referenced/synchronized 1 Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation:	Signal(s) updated:	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Axes: If the machine axis has arrived on the reference cam during reference–point approach, the machine axis is referenced and the IS “Referenced/Synchronized 1” is set. Spindles: After Power On, a spindle is synchronized after a spindle revolution at the latest (360 degrees) (zero mark overrun or BERO responded).	
Signal status 0 or edge change 1 ---> 0	The machine axis/spindle with position measuring system 1 is not referenced/synchronized.	
Related to	IS ”Position measuring system 1“	
References		

Spindle

9

Brief description

Depending on the machine type, the following functions are possible for an analog, NC controlled spindle:

- Setting the direction of rotation of spindle (M3, M4)
- Setting the spindle speed (S)
- Spindle Stop, without orientation (M5)
- Spindle positioning (SPOS=)
  (position–controlled spindle required)
- Gear stage switchover (M40 to M45)
- Thread cutting / tapping (G33, G331, G332, G63)
- Revolutional feed (G95)
- Constant cutting speed (G96)
- Programmable spindle speed limits (G25, G26, LIMS=)
- Position transducer to be mounted either on the spindle or on the spindle motor
- Spindle monitoring for min./max. speed acitve
- Dwell time of spindle in revolutions (G4 S)

Instead of the analog spindle, it is also possible to use a “switched” spindle. In this case, the spindle speed (S) is **not** set from the program but, for example, via manual operation (gearbox) on the machine. It is therefore also not possible to program speed limits. The following can be set from the program:

- Setting the direction of rotation of spindle (M3, M4)
- Spindle Stop, without orientation (M5)
- Tapping (G63)

If this spindle posesses a position transducer, the following additional functions are possible:

- Thread cuttin/tapping (G33)
- Revolutional feed (G95)

When a switched spindle is used, setpoint output for the spindle must be suppressed via machine data (MD: CTRLOUT_TYPE =0).

9.1 Spindle Modes

Spindle modes

An analog spindle controlled by the NC can be operated in three different spindle modes:

- Control mode
- Oscillation mode
- Positioning mode

Spindle mode change

Switchover between the spindle modes is carried out as follows:

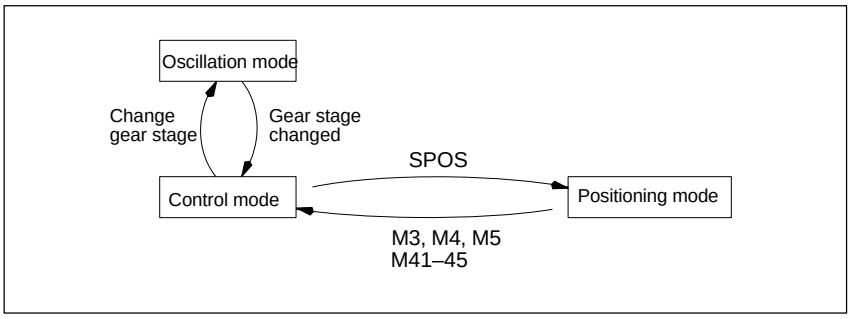


Fig. 9-1 Spindle mode change

- Control mode ---> Oscillation mode
  The spindle changes to oscillation mode if a new gear stage has been set either by automatic gear stage selection (M40) in conjunction with a new S function or by M41 to M45. The spindle will only change to the oscillation mode if the new gear stage is other than the currently active gear stage.
- Oscillation mode ---> control mode
  When the new gear stage is set, the IS “Oscillation mode” is reset, and the mode changes to control mode with the IS “gear stage changed”. The last programmed spindle speed (S function) is active again.
- Control mode ---> positioning mode
  If you wish to stop the spindle from rotation (M3 or M4) with orientation, or you wish to re-orient the spindle from standstill (M5), use SPOS to change to the positioning mode (position–controlled spindle required).
- Positioning mode ---> control mode
  If you wish to end spindle orientation, use M3, M4 or M5 to change to control mode. The last programmed spindle speed (S function) is active again.
- Positioning mode ---> Oscillation mode
  If you wish to end spindle orientation, use M41 to M45 to change to oscillation mode. When the gear stage change is completed, the last programmed spindle speed (S function) and M5 (control mode) are active again.

9.1.1 Spindle Control Mode

When control mode?

The spindle is in control mode when the following functions are active:

- constant spindle speed S, M3/M4/M5 and G94, G95, G97
- constant cutting speed G96 S, M3/M4/M5
- constant spindle speed S, M3/M4/M5 and G33

Preconditions

- Spindle need not be synchronized.
- No spindle actual position encoder required for M3/M4/M5 in conjunction with feed F in mm/min or inch/min (G94).
- A spindle actual position encoder is absolutely necessary for M3/M4/M5 in conjunction with revolutional feed (G95, F in mm/rev. or inch/rev.), constant cutting speed (G96, G97) and tapping (G33).

General spindle reset

The spindle can be stopped by means of the IS ”Delete distance to go/ spindle reset“. CAUTION: When G94 is active, the program execution is continued without any further actions!

Own spindle reset

MD: SPIND_ACTIVE_AFTER_RESET is used to set the behavior of the spindle after Reset or end of program (M2, M30):

- If MD: SPIND_ACTIVE_AFTER_RESET=0, the spindle is immediately decelerated to a standstill with maximum acceleration. The last programmed spindle speed and direction of rotation of the spindle are deleted.
- If MD: SPIND_ACTIVE_AFTER_RESET=1 (own spindle reset) is set, the last programmed spindle speed (S function) and the last programmed direction of rotation of the spindle (M3, M4, M5) are kept.
  If constant cutting speed (G96) is active prior to Reset or end of program, the currently active cutting speed (related to 100 % spindle override) is internally accepted as the last programmed spindle speed.

Own spindle reset

- The spindle override switch is valid.

Note:
A separate spindle override switch is only installed on the machine control panel (MCP) as an option.

- The spindle can always be decelerated by means of the IS ”Clear distance to go/spindle reset”**.**
  CAUTION: When G94 is active, program execution is continued! When G95 is active, the axes are also stopped due to the missing feedrate, and program execution also stops if G1, G2, ... is active.

9.1.2 Spindle Positioning Mode

When positioning mode?

When the programmable function SPOS= ... is active, the spindle in positioning mode.

SPOS= .....

Spindle positioning to an absolute position (0 to 360 degrees) on the shortest way. The direction of positioning is defined either by the current direction of rotation of the spindle (spindle rotates) or automatically by the control system (machine data) (spindel on standstill).

Block change

Programming with SPOS:
Block change is carried out if all functions programmed in the block have reached their end–of–block criterion (e.g. axis traversing completed, all auxiliary functions acknowledged from PLC) **and** the spindle has reached its position (IS "Exact stop fine" for the spindle (V39030000.7)).

Preconditions

- The spindle need not be synchronized.
- Spindle actual position encoder is absolutely necessary.

Positioning from rotation

Positioning from rotation

At the moment when positioning is started (SPOS in the program), the spindle can be in speed–controlled mode. This results in the following sequence:

- Case 1: Spindle in speed–controlled mode, encoder limit frequency range exceeded (Fig. 9-2)
- Case 2: Spindle in speed–controlled mode, encoder limit frequency not exceeded (Fig. 9-3)

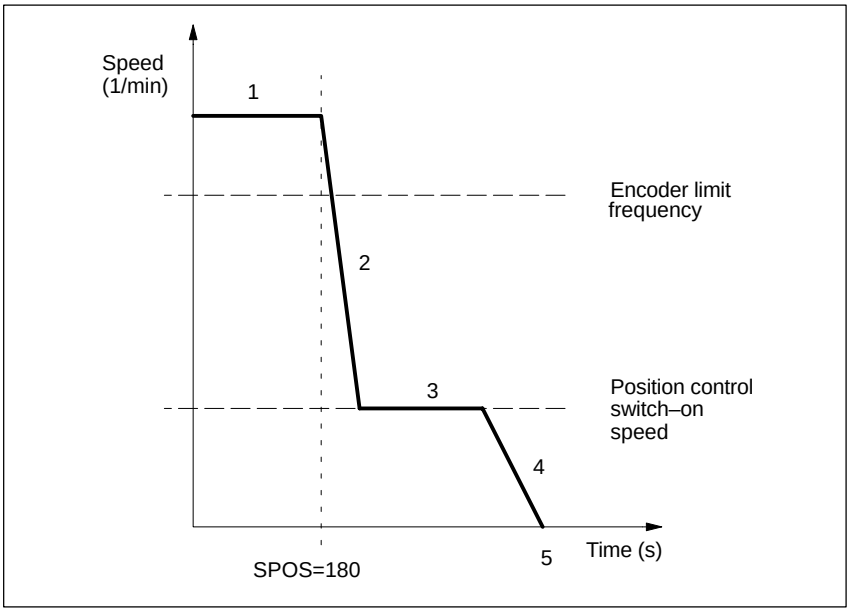


Fig. 9-2 Positioning from rotation, with the programmed spindle speed (and the actual spindle speed) **above** the limit frequency of the spindle actual position encoder (special case).

Spindle speed > limit frequency of encoder

Phase 1:

Spindle rotates with a speed higher than the encoder limit frequency. The spindle is not yet synchronized.

Phase 2:

At the moment when the SPOS command becomes active, the spindle starts deceleration of the spindle to the position controller switch–on speed. When the actual encoder limit frequency is lower than the set encoder limit frequency, the spindle is synchronized. The positioning mode is activated with the synchronization.

Phase 3:

When the position controller switch–on speed set in MD SPIND_POSCTRL_VELO is reached,

- the position control is switched on,
- the distance to go (to the target position) is calculated,
- the acceleration has changed to GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode).

Phase 4:

The spindle decelerates from the calculated ”Brake point” to the target positon with GEAR_STEP_POSCTRL_ACCEL.

Phase 5:

The positon control remains active and keeps the position as programmed. The IS ”Exact stop fine“ and ”Exact stop coarse“ are set if the distance between the spindle position and the programmed position (spindle set position) is less than the exact stop tolerance fine and coarse (defined in the MD: STOP_LIMIT_FINE and MD: STOP_LIMIT_COARSE).

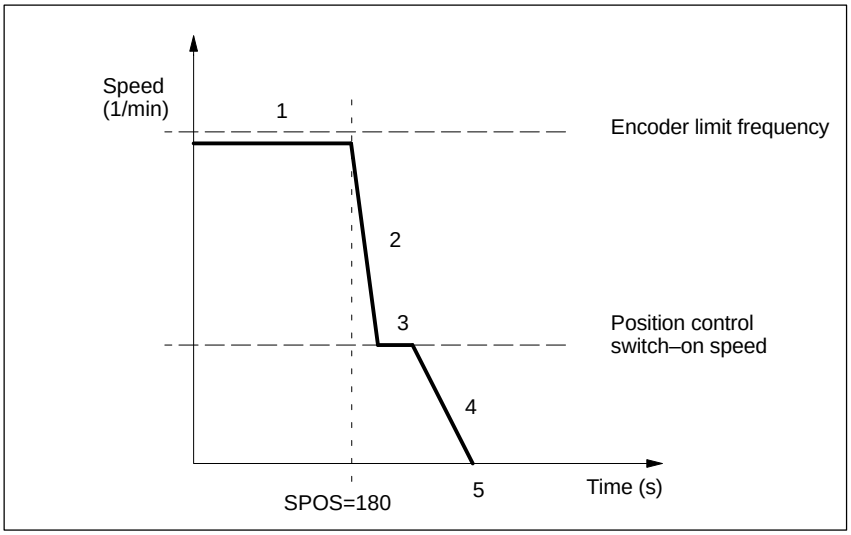


Fig. 9-3 Positioning from rotation, with the programmed spindle speed (and the actual spindle speed) **below** the limit frequency of the spindle actual position encoder (normal case). The position control remains switched off.

Spindle speed less than encoder limit frequency

Phase 1:

Spindle rotates with a speed less than encoder limit frequency. The spindle is synchronized.

Phase 2:

At the moment when the SPOS command becomes active, the spindle starts to decelerate with the acceleration stored in
MD: GEAR_STEP_SPEEDCTRL_ACCEL to the position controller switch–on speed

Phase 3:

When the position controller switch–on speed stored in MD SPIND_POSCTRL_VELO is reached,

- the position control is switched on,
- the distance to go (to the target position) is calculated,
- the acceleration changes to GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode).

Phase 4:

The spindle decelerates with GEAR_STEP_POSCTRL_ACCEL from the calculated “Brake point” to the target position.

Phase 5:

The position control remains active and keeps the spindle on the programmed position. The IS ”Exact stop fine” and “Exact stop coarse” are set if the distance between the spindle position and the programmed position (spindle set position) is less than the exact stop tolerance fine and coarse (defined in the MD: STOP_LIMIT_FINE and MD: STOP_LIMIT_COARSE).

Positioning from standstill

Positioning from standstill

If you wish to position the spindle from standstill, the following two cases are differentiated:

- **Case 1:** The spindle is not synchronized. This is the case if you wish to position the spindle after switching on the control system and the drive.
- **Case 2:** The spindle is synchronized. This is the case if the spindle has been rotated by at least one spindle revolution with M3 or M4 prior to the first positioning and after control system and drive have been switched on and if the spindle has then be stopped with M5 (synchronization with the zero mark).

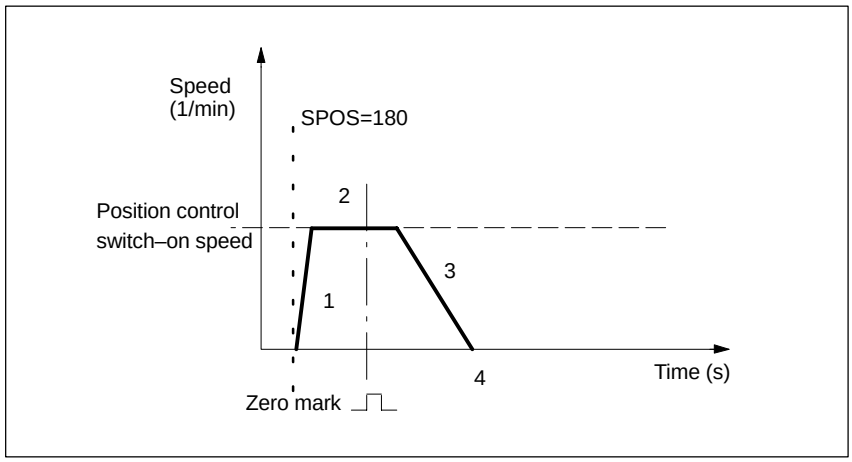


Fig. 9-4 Positioning with the spindle stopped and not synchronized

Case 1: Spindle not synchronized

Phase 1:

When SPOS is programmed, the spindle will accelerate with the acceleration set in MD: GEAR_STEP_SPEEDCTRL_ACCEL (acceleration in speed–controlled mode). The direction of rotation is defined by the
MD: SPIND_POSITIONING_ DIR (direction of rotation when positioning from standstill). The spindle is synchronized with the next zero mark provided from the spindle actual value encoder and then changes to position control mode. A monitoring function checks whether the zero mark is found within the distance defined by MD: REFP_MAX_MARKER_DIST. If the speed entered in
MD: SPIND_POSCTRL_VELO (positioning speed) is reached without synchronizing the spindle, the spindle goes on rotating with the position controller switch–on speed (no more acceleration).

Phase 2:

When the spindle is synchronized, the position control is switched on. The spindle goes on rotating, not exceeding the speed set by MD: SPIND_POSCTRL_VELO, until the brake start point detects the programmed spindle with the defined acceleration.

Phase 3:

At the moment detected by the brake start point calculation in phase 2, the spindle decelerates with the acceleration set in
MD: GEAR_STEP_ POSCTRL_ACCEL (acceleration in position control mode) to standstill.

Phase 4:

The spindle is on standstill and has reached the programmed position. The position control is active and keeps the spindle on the programmed position. The IS ”Position reached with exact stop fine/coarse” is set if the distance between the spindle actual position and the programmed position (spindle set positon) is less than the exact stop tolerance fine and coarse (defined in MD: STOP_LIMIT_FINE and MD: STOP_LIMIT_COARSE).

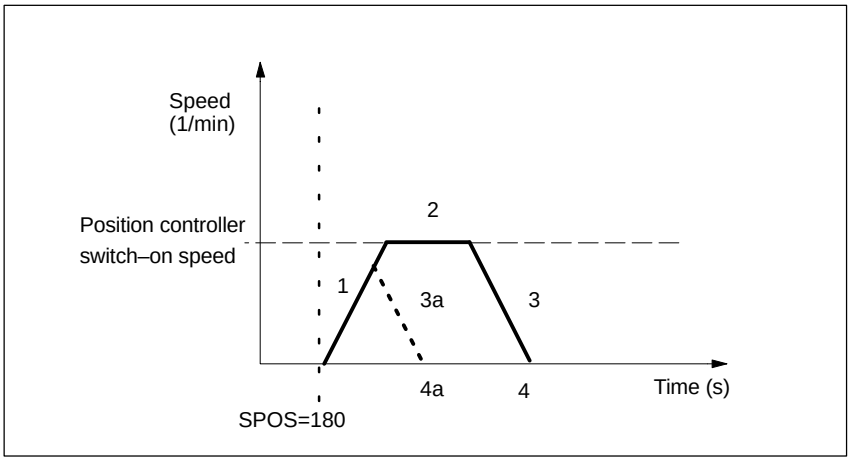


Fig. 9-5 Positioning with the synchronized spindle on standstill

Case 2: Spindle is synchronized

Phase 1:

The spindle is synchronized. When SPOS is programmed, the spindle is switched to position control mode. The acceleration defined in MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode) becomes active. The direction of rotation is defined by the distance to go. The speed entered in MD: SPIND_POSCTRL_VELO (position control switch–on speed) is not exceeded. The calculation of the distance to be traversed to the target position is carried out.

Traversing of the spindle to the programmed target point is carried out in an optimum time, i.e. the target point is approached with maximum possible speed (however, not higher than SPIND_POSCTRL_VELO). Depending on the respective marginal conditions, the phases 1 - 2 - 3 - 4 and 1 - 3a - 4a, respectively, are passed (see Fig. 9-5).

Phase 2:

In order to reach the target point, it was accelerated to the speed entered in MD: SPIND_POSCTRL_VELO (position control switch–on speed). This speed will not be exceed. The brake point start calculation detects the moment when the programmed spindle position can be exactly approached with the acceleration defined in GEAR_STEP_POSCTRL_ACCEL.

At the moment detected by the brake start point calculation in phase 1, the spindle decelerates to standstill with the acceleration defined in MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode).

Phase 3:

At the moment detected by the brake start point calculation in phase 3, the spindle decelerates to standstill with the acceleration defined in MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode).

Phase 3a:

Already at the moment when the SPOS command becomes active, the target point is so close that the spindle can no longer be accelerated to SPIND_POSCTRL_VELO. The spindle will be decelerated to standstill with the acceleration defined in MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode).

Phase 4, 4a:

The spindle is on standstill and has reached its positon. Position control is active and keeps the spindle on its programmed position. The IS ”Position reached with exact stop fine/coarse” are set if the distance between the spindle actual position and the programmed position (spindle set position) is less than the value of the exact stop tolerance fine and coarse (defined in MD: STOP_LIMIT_FINE and MD: STOP_LIMIT_COARSE).

Spindle Reset

The positioning process can be aborted by means of the IS ”Clear distance to go/spindle reset.

The IS “Clear distance to go/spindle reset“ cannot be used to abort the spindle mode ”Positioning mode“.

Distinctive features

- The following machine data are used to define the accelerations:
  MD: SPIND_POSCTRL_ACCEL (acceleration in position control mode)
  MD: SPIND_SPEEDCTRL_ACCEL (acceleration in speed–controlled mode).
- The spindle override switch is valid.
- The positioning (SPOS) is aborted with Reset.
- The positioning is aborted with NC-STOP.

9.1.3 Spindle Positioning Mode

What is meant by oscillation?

When oscillation mode is active, the spindle motor rotates alternating in clockwise direction and counterclockwise direction. This oscillation motion is supported by slight meshing of a new gear stage.

Preconditions

- No spindle actual position encoder required
- Spindle need not be synchronized

Starting oscillation mode

The spindle is in oscillation mode if a new gear stage is set either by automatic gear stage selection (M40) or by M41 to M45 (IS ”Change gear stage“ (V39032000.3) is set). The IS ”Change gear stage” is only set if the new gear stage is other than the currently active gear stage. Oscillation of the spindle is started with the IS ”Oscillation speed” (V38032002.5).

If only the IS “Oscillation speed” is set without setting a new gear stage, no change to oscillation mode is carried out.

Oscillation is started by means of the IS “Oscillation speed”. Depending on the functional sequence and the IS “Oscillation by PLC” (V38032002.4), oscillation differentiates:

- Oscillation by NCK
- Oscillation by PLC

Oscillation time

For each direction of rotation, the oscillation time for oscillation mode can be set in machine data:

- Oscillation time in M3 direction (in the following called t1) in
  MD: SPIND_OSCILL_TIME_CW
- Oscillation time in M4 direction (in the following called t2) in
  MD: SPIND_OSCILL_TIME_CCW

Oscillation by NCK

Phase 1:

When the IS ”Oscillation speed” (V38032002.5) is provided, the spindle motor accelerates to the speed set in MD: SPIND_OSCILL_DES_VELO (oscillation speed) (with oscillation acceleration). The start direction is defined by MD: SPIND_OSCILL_START_DIR (start direction for oscillation). The time t1 (or t2) is started depending on which start direction is defined in MD: SPIND_OSCILL_START_DIR.

Phase 2:

When time t1 (t2) has elapsed, the spindle motor accelerates into the opposite direction to the speed set in MD: SPIND_OSCILL_DES_VELO (oscillation speed). Time t2 (t1) is started.

Phase 3:

When time t2 (t1) has elapsed, the spindle motor accelerates into the opposite direction (same direction as with phase 1) to the speed set in
MD: SPIND_OSCILL_DES_VELO Time t1 (t2) is started. Continuation with phase 2.

Oscillation by PLC

When the IS ”Oscillation speed” is active, the spindle motor accelerates to the speed set in MD: SPIND_OSCILL_DES_VELO (oscillation speed) (with oscillation acceleration). The direction of rotation is defined by means of the IS ”Set direction of rotation CCW“ and the IS ”Set direction of rotation CW”. The oscillation motion and the two times t1 and t2 (time for CW and CCW rotation) must be simulated in the PLC.

End of oscillation mode

The IS ”Gearbox changed” (V38032000.3) informs the NCK that the new gear stage (IS “Actual gear stage”) is valid and oscillation mode is ended. The actual gear stage should correspond to the set gear stage. Oscillation mode is also ended if the IS “Oscillation speed” is still set. The last programmed spindle speed (S function) and direction of rotation of the spindle (M3, M4 or M5) are active again.

When the oscillation mode is ended, the spindle is in control mode again.

All gear–specific limit values (min./max. speed of gear stage, etc.) correspond to the values specified for the actual gear stage and are switched off on standstill of the spindle.

Block change

If the spindle has been changed to oscillation mode (IS ”Change gear stage“ (V39032000.3) is set), part program execution remains stopped. No new block is started for execution. If the oscillation mode is ended with the IS ”Gear stage changed” (V38032000.3), part program execution is continued as shown in Fig. 9-6. Any new blocks will be processed.

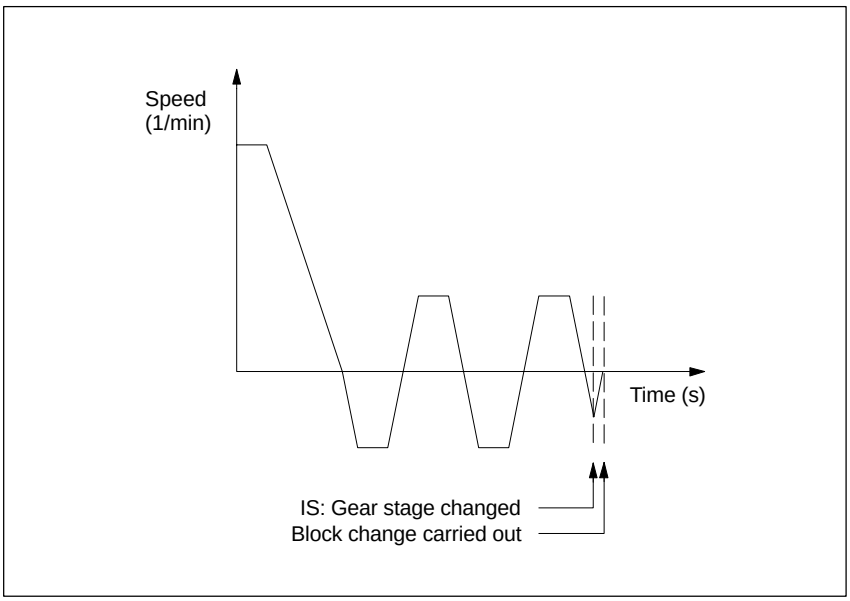


Fig. 9-6 Block change after oscillation mode

Spindle Reset

The spindle can be decelerated by means of the IS ”Clear distance to go/spindle reset” (V30000000.7). After standstill, the spindle is in control mode. The S word is deleted, M5 activated.

Distinctive features

- The acceleration is defined in MD: SPIND_OSCILL_ACCEL (acceleration during oscillation).
- If the IS ”Oscillation speed (V38032002.5) is reset, the oscillation motion will stop. However, the spindle oscillation mode will not be left.
- The spindle override switch is disabled (fixed to 100%). An exception is only the position 0%.
- The IS ”Reset” (V30000000.7) will not abort oscillation mode.
- When indirect measuring systems are used, the synchronization gets lost.

Reset during gear stage change

No spindle stop is possible due to

- IS ”Reset” (V30000000.7)
- IS ”NC Stop” (V32000007.3)

if

- the spindle is in oscillation mode for gear stage change
- the IS ”Gear stage changed” (V38032000.3) is not yet present.

In these cases, when Reset is selected, the alarm 10640 ”Stop during gear stage change not possible”.
After the gear stages have been changed, the Reset request is carried out and the alarm deleted if this is still present on the interface.

Note

Only possibility of abortion:
IS ”Clear distance to go/spindle reset” (V38030002.2) (own spindle reset) active.

9.2 Referencing/Synchronizing

Why synchronizing?

In order to ensure that the control system finds the 0 degree position after switching on, the CNC must be synchronized with the position measuring system of the spindle. This process is called synchronization.

Only a synchronized spindle can perform:

- thread cutting
- positioning

Why referencing?

Axes are synchronized via reference–point approach. This process is also called “referencing” (see Section “Reference–Point Approach”).

Synchronization sequence

After the control system has been switched on, the spindle can be synchronized as follows:

- The spindle is started with a spindle speed (S function) and a direction of rotation of the spindle (M3 or M4) and synchronizes itself with the next zero mark of the position measuring system.
- From standstill, the spindle is positioned with SPOS. The spindle accelerates to the positioning speed and synchronizes itself with the next zero mark of the position measuring system. The spindle is then positioned to the programmed position.

Note

During synchronization of the spindle, the reference–point value and the offset of the reference point are enabled.

Max. encoder frequency exceeded

If the spindle operated in spindle control mode reaches a speed (high S value programmed) higher than the max. encoder limit frequency (the max. speed of the encoder may not be exceeded), the synchronization gets lost. The spindle goes on rotating, but with reduced functionality.

The following functions will reduce the spindle speed as long as the active measuring system is below the encoder limit frequency:

- Thread cutting (G33)
- Revolutional feed (G95)
- Constant cutting speed (G96, G97)

If the speed is below the max. encoder limit frequency (smaller S value programmed, spindle override switch modified, etc.), the spindle automatically synchronizes itself with the next zero mark or the next BERO signal.

If the encoder limit frequency is exceeded, the IS “Referenced/synchronized 1” (V39030000.4) is reset and the IS ”Encoder limit frequency 1 exceeded” (V39030000.2) set.

Resynchronizing

In the following cases, the position measuring system of the spindle has to be resynchronized with the 0–degree position:

- The position encoder is mounted on the motor, a BERO on the spindle, and gear stage change is carried out. The synchronization is triggered internally if the spindle is in the new gear stage (see Synchronization Sequence).

9.3 Speed and Gear Stage Change

Speeds

Data for 5 gear stages can be entered into the control system.

The gear stages are defined by a minimum speed and a maximum speed for the gear stage and by a minimum speed and a maximum speed for automatic gear stage change.

The new gear stage is only output if the newly programmed speed setpoint cannot be realized with the current gear stage.

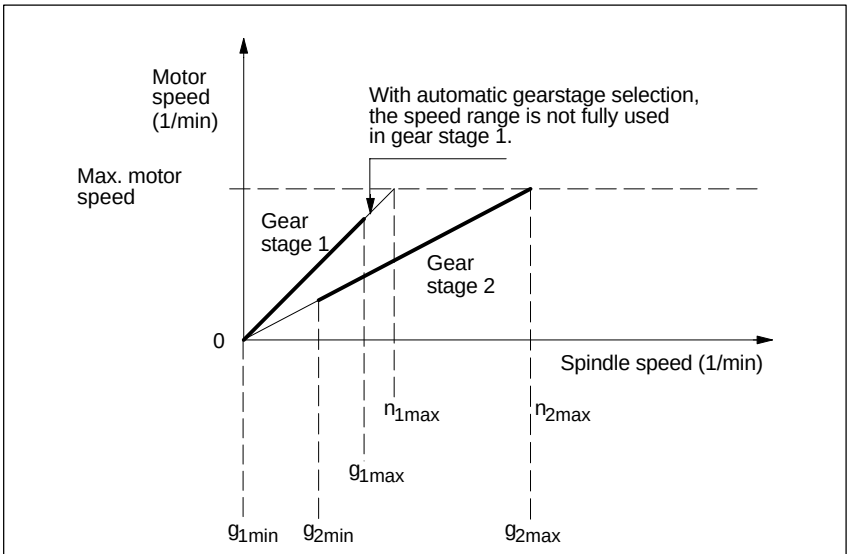
For simplification, the oscillation times for gear stage change can be set directly in the 802S; otherwise, the oscillation function must be realized in the PLC. The oscillation function is triggered by the PLC.

Why gear stages?

Gear stages for the spindle are used to step down the motor speed, thus generating a high torque with low spindle speeds.

Number of gear stages

5 gear stages can be projected for each spindle. If the spindle motor is mounted directly (1:1) on the spindle or with a non–variable gear ratio, the MD: GEAR_STEP_CHANGE_ENABLE (gear stage change possible) must be set to zero.



The following can be set via MD:

• n_{1max} ...	Max. spindle speed of 1st gear stage
• g_{1min} ...	Min. spindle speed of 1st gear stage for autom. gear stage selection
• g_{1max} ...	Max. spindle speed of 1st gear stage for autom. gear stage selection
• n_{2max} ...	Max. spindle speed of 2nd gear stage
g_{2min} ...	Min. spindle speed of 2nd gear stage for autom. gear stage selection
• g_{2max} ...	Max. spindle speed of 2nd gear stage for autom. gear stage selection

Fig. 9-7 Gear stage change with gear stage selection

Preselecting the gear stage

Gear stages can be preset:

- fixed by the part program (M41 to M45)
- automatically by the programmed spindle speed (M40)

When M40 is used for automatic gear stage selection with an S word, the spindle must be in spindle control mode. Otherwise, the gear stage change will be refused and alarm 22000 set.

M41 to M45

The gear stage can be specified as a fixed value by means of M41 to M45. If a gear stage other than the current (actual) gear stage is set, the IS "Change gear stage" (V39032000.3) and the IS "Set gear stage A to C" (V39032000.0 to .2) are set. The programmed spindle speed (S function) will then refer to this fixed gear stage. If a spindle speed above the max. speed of the fixed gear stage is programmed, a limitation to the max. speed of the gear stage is carried out and the IS "Set speed limited" (V39032001.1) set.

M40

M40 in the part program is used by the control system to set the gear stage automatically. A check is performed to see in which gear stage the programmed spindle speed (S function) is possible. If a gear stage is found which is other than the current (actual) gear stage, the IS “Change gear stage” (V39032000.3) and the IS ”Set gear stage A to C” (V39032000.0 to .2) are set.

The automatic gear stage selection is carried out such that the programmed spindle speed is first compared with the min. and max. speed of the current gear stage. If the comparison ends with a positive result, no new gear stage is set. In case of a negative result, the comparison (starting with gear stage 1) is carried out for all 5 gear stages as long as it ends with a positive result. If the comparison is even in the 5th gear stage not positive, no gear stage change is carried out. The speed is either limited to the max. speed of the current gear stage (if necessary) or increased to the minimum speed of the current gear stage and the IS ”Set speed limited“ (V39032001.1) or ”Set speed increased” (V39032001.2) set.

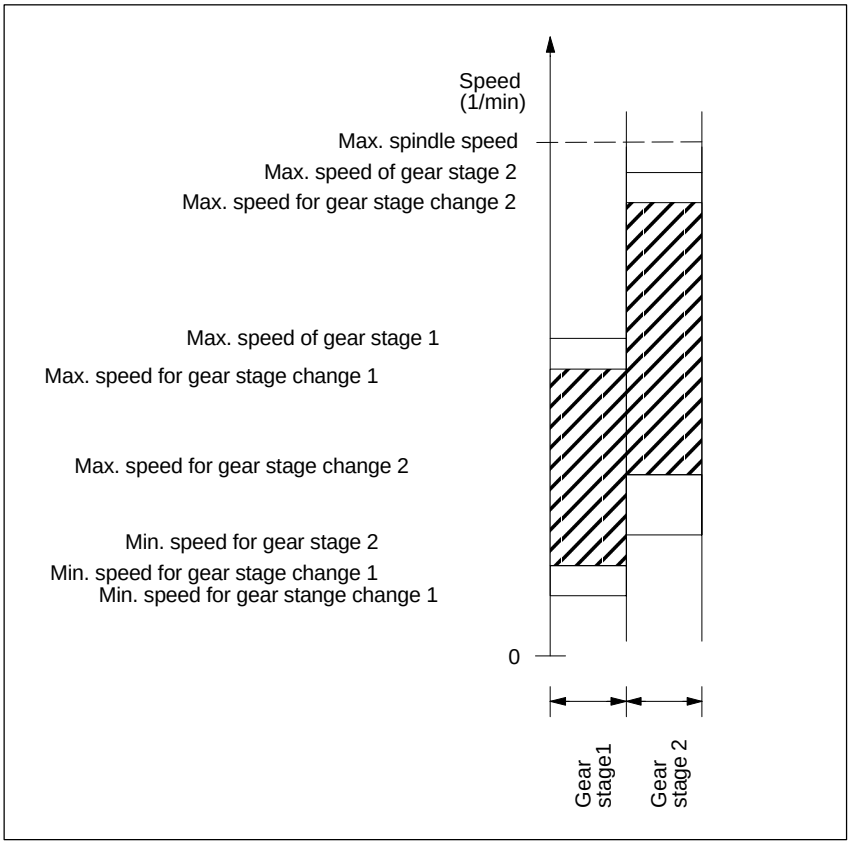


Fig. 9-8 Example of speed ranges with automatic gear stage selection (M40)

Gear stage change with the spindle on standstill

IWhen the new gear stage is selected by M40 and spindle speed or M41 to M45, the IS ”Set gear stage A to C” (V39032000.0 to .2) and IS ”Change gear stage“ (V39032000.4) are set. Depending on at which moment the IS ”Oscillation speed” (V38032002.5) is set, the spindle decelerates to standstill with the acceleration for oscillation or with the acceleration for speed–controlled mode/positon control mode.

The next block in the part program after gear stage change by M40 and S value or M41 to M45 will not be executed (same effect as the IS “Read–in disable” (V32000006.1) would be set).

Oscillation is switched on at the latest with the standstill of the spindle (IS ”Axis/spindle on standstill“ (V39030001.4)) using the IS “Oscillation speed” (V38032002.5). When the new gear stage is active, the PLC user sets the IS ”Actual gear stage“ (V38032000.0 to .2) and ”Gear stage changed“ (V38032000.3). The gear stage change is considered completed (spindle mode “Oscillation” is deselected) and a change to the parameter record of the new actual gear stage is carried out. In the new gear stage, the spindle accelerates to the last programmed spindle speed. The next block in the part program can be executed. The IS “Change gear stage” (V39032000.3) is reset by the NCK, and as a response, the PLC user will reset the IS “Gear stage changed” (V38032000.3).

Parameter record

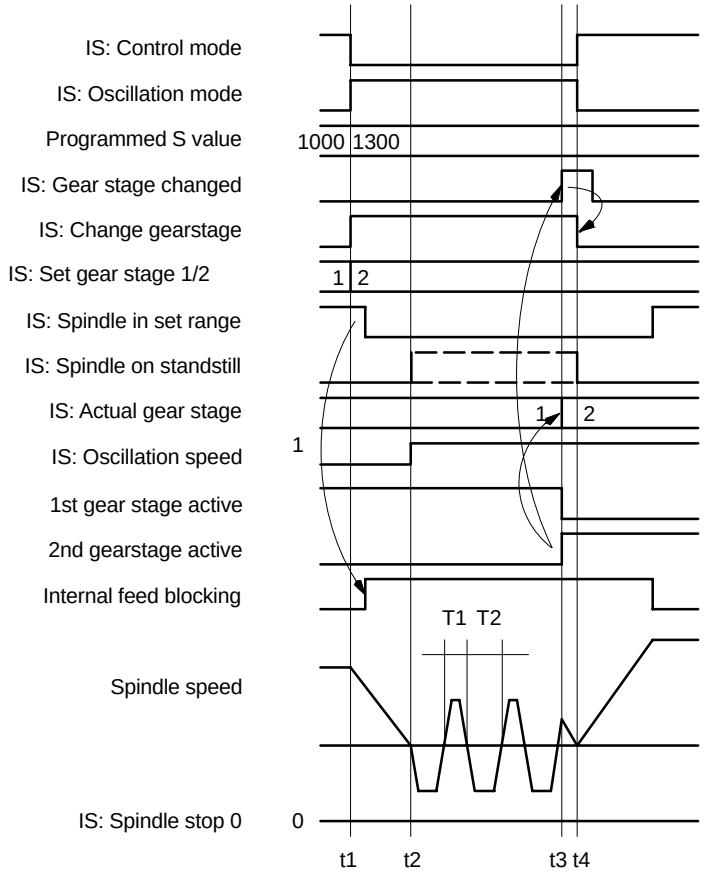
Each of the 5 gear stages is assigned a parameter record with the following assignment:

Parameter Record No.	PLC Interface	Data of the Data Block	Contents
0	-	Data for axis mode	K_V factor (loop gain) Monitoring functions M40 speed Min./max. speed ....
1	000 001	Data for 1st gear stage	
2	010	Data for 2nd gear stage	
3	011	Data for 3rd gear stage	
4	100	Data for 4th gear stage	
5	101 110 111	Data for 5th gear stage - -	

Distinctive features

To decelerate the spindle, the PLC user need not set the IS ”Spindle Stop“ (V38030004.3). The IS ”Spindle Reset” (V38030002.2) aborts gear stage change. At the same time, the programmed spindle speed and the programmed direction of rotation of the spindle are deleted. After gear stage change, the spindle will not accelerate to the programmed spindle speed.

Typical time sequence for gearchange with the spindle on standstill:



t1 When S1300 is programmed, the NCK detects a new gear stage (2nd gear stage), sets the IS ”Change gear stage” and blocks execution of the next part program block.

t2 The spindle is on standstill and oscillation (oscillation by NCK) is started. The IS ”Oscillation speed” must be set on time t2 at the latest.

t3 The new gear stage is active. The PLC user transfers the new (actual) gear stage to the NCK and sets the IS ”Gear stage changed”.

t4 As a response, the NCK receives the IS ”Change gear stage”, ends oscillation, enables the next part program block for exection and accelerates the spindle to the new S value.

Fig. 9-9 Gear stage change with the spindle at a standstill

9.4 Programming

The spindle can be dimensioned for the following programmable functions:

- G95 Revolutional feedrate
- G96 S... LIMS=... Constant cutting rate in m/min, upper limit speed
- G97 Cancel G96 and freeze last spindloe speed
- G33 Thread cutting
- G331, G332 Thread interpolation
- G25 S..., G26 S... Programmable lower/upper speed limit
- G4 S... Dwell time in spindle revolutions
- Programming of

  M3 Direction of rotation of spindle CW
  M4 Direction of rotation of spindle CCW
  M5 Spindle Stop, without orientation
  S... Spindle speed in 1/min , e.g.: S300
  SPOS=... Spindle positioning, e.g.: SPOS=270
  M40 Automatic gear stange selection for the spindle
  M41 to M45 Select gear stage 1 to 5 for the spindle

9.5 Spindle Monitoring

Speed ranges

Both the spindle monitoring functions and the currently active functions (G94, G95, G96, G33, etc.) define the permissible speed ranges of the spindle.

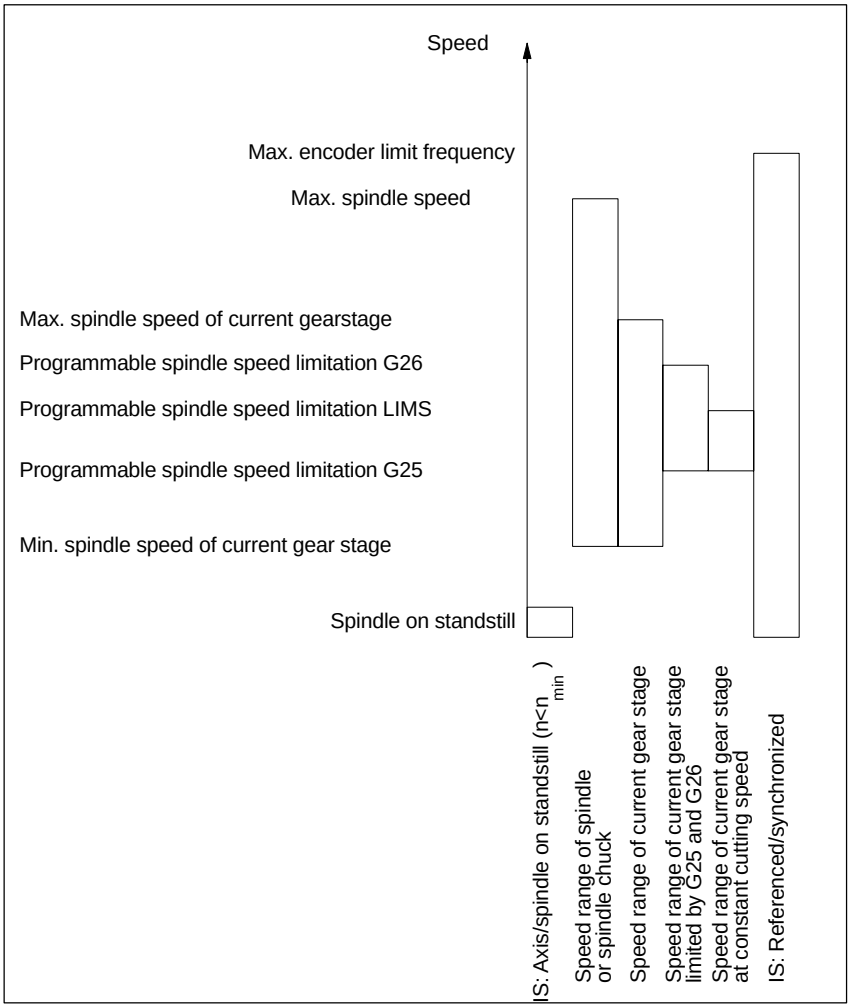


Fig. 9-10 Ranges of spindle monitoring functions / speeds

9.5.1 Axis/Spindle on Standstill ($n<n_{min}$)

Only if the axis/spindle is on standstill**,** i.e. the spindle actual speed is below a value which can be set in MD: STANDSTILL_VELO_TOL certain functions, such as tool change, open machine door, enable path feed, etc., are possible.

- If the spindle is at a standstill, the IS “Axis/spindle stopped“ (V39030001.4) is set.

The monitoring function is active in the three spindle modes.

9.5.2 Spindle in Set Range

Function

The spindle monitoring function ”Spindle in set range” checks whether the programmed spindle speed is reached, the spindle is on standstill (IS “Axis/spindle on standstill”) or still in the acceleration phase.

In the spindle control mode, the set speed (programmed speed * spindle override with consideration of the active limits) is compared with the actual speed. If the actual speed deviates from the set speed by more than the spindle speed tolerance (MD: SPIND_DES_VELO_TOL (spindle speed tolerance)):

- the IS ”Spindle within set range“ (V39032001.5) is set to zero;
- path feed is blocked internally by the NCK.

9.5.3 Max. Spindle Speed

Max. spindle speed

For spindle monitoring ”Max. spindle speed”, a max. speed is defined which may not be exceeded by the spindle. The max. spindle speed is entered in MD: SPIND_MAX_VELO_LIMIT. The NCK will limit the set spindle speed to this value. If the spindle actual speed nevertheless exceeds the max. spindle speed with consideration of the spindle speed tolerance (MD: SPIND_DES_VELO_TOL (spindle speed tolerance)), a drive error is present and the IS “Speed limit exceeded” (V39032002.0) is set. In addition, alarm 22100 is output and all axes and the spindle are decelerated.

Speed limitation from PLC

The spindle speed can be limited to a certain value by means of the PLC: This value is stored in MD: SPIND_EXTERN_VELO_UNIT and activated via the IS ”Velocity/spindle speed limitation“ (V38030003.6).

9.5.4 Min./Max. Speed of Gear Stage

Max. speed

The max. speed of the gear stage is entered in MD: GEAR_STEP_MAX_VELO_LIMIT. This (set) speed can never be exceeded in the activated gear stage. When the programmed spindle speed is limited, the IS “Set speed limited” (V39032001.1) is set.

Min. speed

The min. speed of the gear stage is entered in MD: GEAR_STEP_MIN_VELO_LIMIT. The actual speed can never be below the (set) speed even if a too low S value is programmed. The interface signal ”Set speed increased” (V39032001.2) is set.

The min. speed of the gear stage is only active in speed–controlled mode and can only fall below the set speed in the following cases:

- Spindle override 0%
- M5
- S0
- IS ”Spindle Stop“
- IS ”Servo enable” canceled
- IS ”Reset”
- IS ”Spindle Reset“
- IS ”Oscillation speed”
- ”NC STOP for axis/spindle“
- IS ”Axis/spindle blocked“

9.5.5 Max. Encoder Limit Frequency

Warning

The max. encoder limit frequency of the spindle actual position encoder is monitored by the control system (exceeding possible). The machine manufacturer must ensure by appropriate design of the components spindle motor, gearbox, resolver gearbox and the associated machine data, that the max. speed (mechanical limit speed) of the spindle actual position encoder cannot be exceeded.

Max. encoder limit frequency exceeded

If the spindle in spindle control mode or oscillation mode reaches a speed (high S value programmed) which is above the max. encoder limit frequency (the max. mechanical limit speed of the encoder may not be exceeded), the synchronization gets lost. However, the spindle goes on rotating.

When one of the functions

- thread cutting (G33)
- tapping without compensating chuck (G331, G332)
- revolutional feed (G95)
- constant cutting speed (G96, G97)

is programmed, the spindle speed is automatically reduced to a value at which the active measuring system operates properly again.

If no measuring system is connected (MD: NUM_ENC = 0), the speed actual value is internally derived from the speed setpoint and displayed.

Max. encoder limit frequency undershot

If the max. encoder limit frequency has been exceeded and then a speed is reached again which is below the value defined in MD: ENC_FREQ_LIMIT_LOW (smaller S value programmed, spindle override switch changed, etc.), the spindle automatically synchronizes itself with the next zero mark or the next BERO signal.

Distinctive features

When the following functions are active, the max. encoder limit frequency cannot be exceeded:

- Spindle operating mode “Positioning mode”
- Thread cutting (G33)
- Tapping without compensating chuck (G331, G332)
- Revolutional feed (G95)
- Constant cutting speed (G96)

9.5.6 Target Position Monitoring

Function

During positioning (spindle is in spindle positioning mode), a monitoring function checks how far the spindle (i.e. its actual positon) is away from the programmed spindle position (target position).

To this aim, two incremental values can be entered in MD: STOP_LIMIT_COARSE (excat stop limit coarse) and MD: STOP_LIMIT_FINE (exact stop limit fine) as an incremental path (from the spindle set position). Irrespective of the two limit values, the spindle positioning accuracy is always as good as set by the connected spindle encoder, backlash, gearbox ratio, etc.

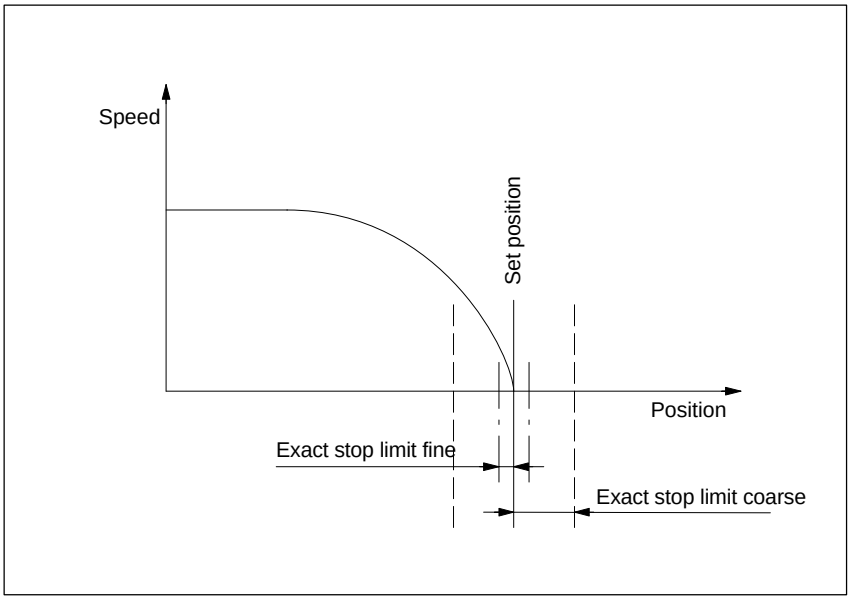


Fig. 9-11 Exact stop ranges of a spindle

IS: Position reached with exact stop

The two limit values defined by the MD: STOP_LIMIT_COARSE and MD: STOP_LIMIT_FINE (exact stop coarse and fine) are output to the PLC with the IS ”Position reached with exact stop coarse“ (V39000000.6) and the IS ”Position reached with exact stop fine“ (V39000000.7).

Block change at SPOS

When SPOS is used to position the spindle, the block change is carried out depending on the target position monitoring with the IS “Position reached with exact stop fine”. The remaining functions programmed in the block must also have reached their end–of–block criterion (e.g. axes ready, all auxiliary functions acknowledged from the PLC).

9.6 Data Description

Machine data

35010 MD number	GEAR_STEP_CHANGE_ENABLE Gear stage change possible		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	If the spindle motor is mounted directly (1:1) on the spindle or with non–variable gear ratio, the MD: GEAR_STEP_CHANGE_ENABLE (gear stage change is possible) must be set to zero. Gear stage change with M40 to M45 is then not possible. If the spindle motor is mounted on the spindle via a gearbox with interchangeable gearboxes, the MD: GEAR_STEP_CHANGE_ENABLE must be set to one. The gearbox can have up to 5 gear stages, which can be selected by M40 to M45.		
Related to	MD: GEAR_STEP_MAX_VELO (max. speed for gear stage change) MD: GEAR_STEP_MIN_VELO (min. speed for gear stage change) The MD: GEAR_STEP_MAX_VELO and MD: GEAR_STEP_MIN_VELO must cover the entire speed range.		

35040 MD number	SPIND_ACTIVE_AFTER_RESET Spindle active after RESET		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	The MD: Spindle active after RESET (SPIND_ACTIVE_AFTER_RESET) is used to set the response of the spindle after Reset and end of program (M2, M30). It is only effective in spindle control mode. MD: SPIND_ACTIVE_AFTER_RESET = 0: Control mode: - Spindle stops - Program aborted Oscillation mode: - Alarm 10640 ”Stop during gear stage change not possible“ - Oscillation is not aborted - Axes are stopped - Program is aborted after gear stage change or Spindle Reset, alarm is deleted Positioning mode: - is stopped MD: SPIND_ACTIVE_AFTER_RESET= 1: Control mode: - Spindle does not stop - Program is not aborted Oscillation mode: - Alarm 10640 ”Stop during gear stage change not possible“ - Oscillation is not aborted - Axes not stopped - Program is aborted after gear stage change, the alarm is deleted and the spindle goes on rotating with the programmed M and S values Positioning mode: - is stopped The IS ”Clear distance to go/Spindle Reset” (V38030001.2) is always active irrespective of MD: SPIND_ACTIVE_AFTER_RESET.		
MD not applicable	to spindle modes other than control mode		
Related to	IS ”Reset” (V30000000.7) IS ”Clear distance to go/spindle reset” (V38030001.2)		

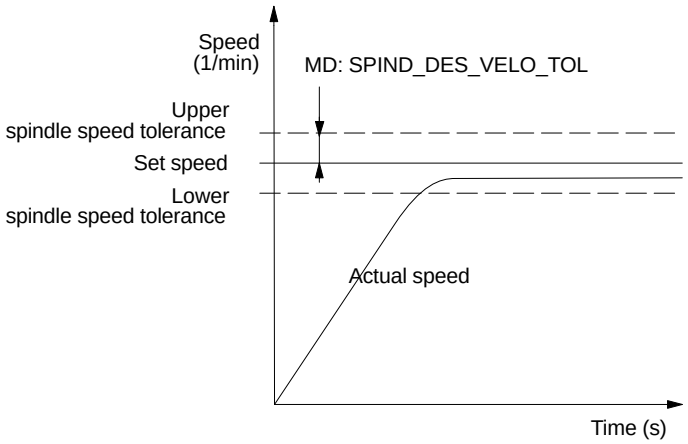
35100 MD number	SPIND_MAX_VELO_LIMIT Max. spindle speed		
Default: 10 000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In this MD, the max. spindle speed is entered which must not be exceeded by the spindle (spindle chuck with workpiece or the tool). The NCK limits the spindle speed to this value. If the max. spindle speed with consideration of the spindle speed tolerance (MD: SPIND_DES_VELO_TOL) is nevertheless exceeded, a drive error is present and the IS "Speed limit exceeded" (V39032001.0) is set. In addition, alarm 22050 "Maximum speed reached" is output and all axes and spindles of the channel decelerated (precondition: encoder is still operative).		
Related to	MD: SPIND_DES_VELO_TOL (spindle speed tolerance) IS "Speed limit exceeded" (39032001.0) Alarm 22050 "Maximum speed reached"		

35110 MD number	GEAR_STEP_MAX_VELO[n] Max. speed for gear stage change [gear stage number]: 0...5		
Default: 500, 500, 1000, 2000, 4000, 8000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In MD: GEAR_STEP_MAX_VELO, the max. speed of the gear stage required for automatic gear stage change (M40) is entered. The gear stages must be defined by MD: GEAR_STEP_MAX_VELO and MD: GEAR_STEP_MIN_VELO such that there are no gaps in the programmed spindle speed range between the gear stages. **Wrong** GEAR_STEP_MAX_VELO [gear stage] =1000 GEAR_STEP_MIN_VELO [gear stage] =1200 **Correct** GEAR_STEP_MAX_VELO [gear stage] =1000 GEAR_STEP_MIN_VELO [gear stage] =950		
Related to	MD: GEAR_STEP_MIN_VELO (min. speed for gear stage change) MD: GEAR_STEP_CHANGE_ENABLE (gear stage change possible) MD: GEAR_STEP_MIN_VELO_LIMIT (min. speed of gear stage) MD: GEAR_STEP_MAX_VELO_LIMIT (max. speed of gear stage)		

35120 MD number	GEAR_STEP_MIN_VELO[n] Min. speed for gear stage change [gear stage number]: 0...5		
Default: 50, 50, 400, 800, 1500, 3000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In MD: GEAR_STEP_MIN_VELO, the min. speed of the gear stage for automatic gear stage change (M40) is entered. For further description see MD: GEAR_STEP_MAX_VELO.		
Related to	MD: GEAR_STEP_MAX_VELO (max. speed for gear stage change) MD: GEAR_STEP_CHANGE_ENABLE (gear stage change is possible) MD: GEAR_STEP_MIN_VELO_LIMIT (min. speed of gear stage) MD: GEAR_STEP_MAX_VELO_LIMIT (max. speed of gear stage)		

35130 MD number	GEAR_STEP_MAX_VELO_LIMIT[n] Max. speed of gears step [gear stage number]; 0...5		
Default: 500, 500, 1000, 2000, 4000, 8000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In MD: GEAR_STEP_MAX_VELO_LIMIT, the max. speed of the gear stage is entered. This speed can never be exceeded in the active gear stage.		
Special cases, errors, ...	• With position control switched on, the value is limited to 90 % (control reserve) • If the programmed S value is above the max. gear stage, the set speed is limited to the max. speed of the currently active gear stage (with gear stage selection - M41 to M45); in addition, the IS: "Programmed speed too high" is set. • If the programmed S value is above the max. speed for gear stage change, a new gear step is set (with automatic gear stage selection - M40). • If the programmed S value is above the max. speed of the highest gear stage, the speed is limited to the max. speed of the highest gear stage (with automatic gear stage selection - M40). • If the programmed S value has no matching gear stage, no gear stage change is triggered.		
Related to	MD: GEAR_STEP_MAX_VELO (max. speed for gear stage change) MD: GEAR_STEP_MIN_VELO (min. speed for gear stage change) MD: GEAR_STEP_CHANGE_ENABLE (gear stage change is possible) MD: GEAR_STEP_MIN_VELO_LIMIT (min. speed of gear stage)		

35140 MD number	GEAR_STEP_MIN_VELO_LIMIT[n] Min. speed of gear stage [gear stage number]: 0...5		
Default: 5, 5, 10, 20, 40, 80	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In MD: GEAR_STEP_MIN_VELO_LIMIT, the min. speed of the gear stage is entered. Even if the programmed S value is too low, the speed cannot be below this speed. The minimum speed can only not be reached by the signals/commands/states mentioned in the Section "Min./max. speed of gear stage".		
MD not applicable to	• Spindle mode "Oscillation mode" • Spindle mode "Positioning mode, axis mode"		
Application example(s)	Smooth running of motor can no longer be ensured below the minimum speed.		
Related to	MD: GEAR_STEP_MAX_VELO (max. speed for gear stage change) MD: GEAR_STEP_MIN_VELO (min. speed for gear stage change) MD: GEAR_STEP_CHANGE_ENABLE (gear stage change is possible) MD: GEAR_STEP_MAX_VELO_LIMIT (max. speed of gear stage)		

35150 MD number	SPIND_DES_VELO_TOL Spindle speed tolerance		
Default: 0,1	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	Protection level: 2/7	Unit: factor	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In the spindle operating mode “Control mode”, the set speed (programmed speed x spindle override with consideration of the limits) is compared with the actual speed. ● If the actual speed deviates from the set speed by more than the spindle speed tolerance (MD: SPIND_DES_VELO_TOL), the IS ”Spindle within set range” (V39032001.5) is set to zero. ● If the actual speed deviates from the set speed by more than the spindle speed tolerance (MD:SPIND_DES_VELO_TOL), the path feed will be blocked. ● If the actual speed exceeds the max. spindle speed (MD: SPIND_MAX_VELO_ LIMIT) by more than the spindle speed tolerance (MD:SPIND_DES_VELO_TOL), the IS ”Speed limit exceeded“ (V39032001.0) is set and alarm 22050 ”Max. speed reached” is output. All axes and spindles of the channel are decelerated.		
MD not applicable	to spindle mode “Oscillation” to spindle mode “Positioning”		
Fig. 9-12			
Related to	MD: SPIND_ON_SPEED_AT_IPO_START (feed enable for spindle within set range) MD: SPIND_MAX_VELO_LIMIT (max. spindle speed) IS ”Spindle within set range” (V39032001.5) IS ”Speed limit exceeded“ (V39032001.0) Alarm 22050 ”Max. speed reached“		

35160 MD number	SPIND_EXTERN_VELO_LIMIT Spindle speed limitation from PLC		
Default: 1000	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In MD: SPIND_EXTERN_VELO_UNIT, a limit value for the spindle speed is entered which is considered exactly at the moment when the IS ”Velocity/speed limitation“ (V38030003.6) is set. The NCK limits the spindle speed to this value.		

35200 MD number	GEAR_STEP_SPEEDCTRL_ACCEL[n] Acceleration in speed–controlled mode [gear stage number]: 0...5		
Default: 30, 30, 25, 20, 15, 10	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: Rev./s^2	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The torque of the spindle in the lower speed range is constant and reduced from a certain defined speed (upper speed range). The lower speed range with the constant torque ends at a speed which must be entered in MD: ACCEL_REDUCTION_SPEED_POINT (speed limit reduced acceleration). If the spindle is in **speed–controlled mode**, the acceleration is entered in the lower speed range (constant torque) in MD: GEAR_STEP_SPEEDCTRL_ ACCEL.		
Special cases, errors, ...	The acceleration in speed–controlled mode (MD: GEAR_STEP_SPEEDCTRL_ ACCEL) can be set higher than in position–control mode (MD: GEAR_STEP_ POSCTRL_ACCEL (acceleration in position control mode)), since no position control reserve must be considered.		
Related to	MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position–control mode) MD: ACCEL_REDUCTION_SPEED_POINT (speed limit reduced acceleration)		

35210 MD number	GEAR_STEP_POSCTRL_ACCEL[n] Acceleration in position control mode [gear stage number]: 0...5		
Default: 30, 30, 25, 20, 15, 10	Min. input limit: 0	Max. input limit: plus	
Changes effective after Power On	Protection level: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	The acceleration in positon control mode must be set such that the current limit is not reached.		
Related to	MD: GEAR_STEP_SPEEDCTRL_ACCEL MD: ACCEL_REDUCTION_SPEED_POINT		

35300 MD number	GEAR_STEP_POSCTRL_ACCEL[n] Position control threshold speed		
Default: 500	Min. input limit: 0	Max. input limit: ***	
Change valid after Power On	Schutzstufe: 2/7	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	When positioning a spindle not being in position control mode, the position control is only connected when the spindle has reached the speed defined in MD SPIND_POSCTRL_VELO. For the behavior of the spindle under various supplementary conditions (positioning from the movement, positoning from the standstill), please refer to the Section “Spindle Mode ’Positioning Mode’”.		
Related to	MD: SPIND_POSITIONING_DIR (direction of rotation when positioning from the standstill if no synchronization is provided		

35350 MD number	SPIND_POSITIONING_DIR Direction of rotation when positioning from the standstill without reference		
Default: 3	Min. input limit: 3	Max. input limit: 4	
Changes effective after Power On	Protection level: 2/7	Unit: -	
Data type: BYTE	Valid as from SW version:		
Meaning:	When SPOS is programmed, the spindle changes to position control mode and accelerates with the acceleration defined by MD: GEAR_STEP_POSCTRL_ACCEL (acceleration in position control mode) if no synchronization has been carried out. The direction of rotation is defined by the MD: SPIND_POSITIONING_DIR (direction of rotation on positioning from standstill). MD: SPIND_POSITIONING_DIR = 3 ---> CW direction of rotation MD: SPIND_POSITIONING_DIR = 4 ---> CCW direction of rotation		
Related to	MD: SPIND_POSCTRL_VELO (position–control switch–on speed)		

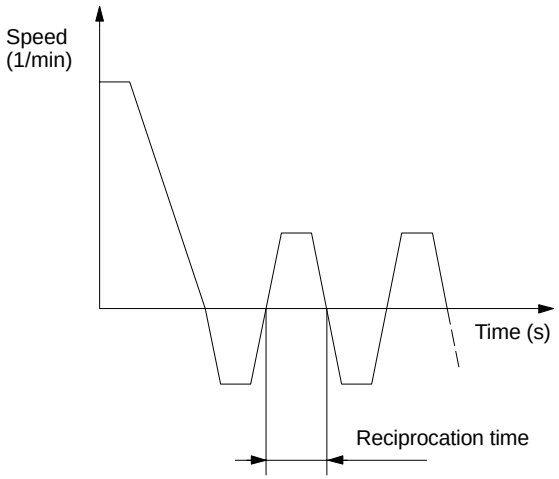
35400 MD number	SPIND_OSCILL_DES_VELO Reciprocation speed		
Default: 500	Min. input limit: 0	Max. input limit: Value in MD: GEAR_STEP_MAX_VELO_LIMIT	
Change effective after Power On	Protection level: 2/7	Unit: rpm	
Data type: DOUBLE	Applies from SW version:		
Meaning:	When reciprocating using the IS “Reciprocation speed” (V38032002.5) a motor speed for the spindle motor has been specified. This motor speed is defined in MD: SPIND_OSCILL_DES_VELO. The motor speed defined in this MD is independent of the current gear stage. In AUTOMATIC and MDA screen forms, the reciprocation speed is displayed in the Set Spindle window until the gear change is completed.		
MD not applicable	to spindle modes other than reciprocation mode		
Application example(s)	Engaging of a new gear stage can be facilitated by reciprocating of the spindle motor, since the gear wheels can thus better be meshed.		
Special cases, errors, ...	For the reciprocation speed defined in this MD, the acceleration when reciprocating applies (MD: SPIND_OSCILL_ACCEL).		
Related to	MD: SPIND_OSCILL_ACCEL (acceleration when reciprocating) IS ”Reciprocation by PLC” (V38032002.4) IS ”Reciprocation speed” (V38032002.5)		

35410 MD number	SPIND_OSCILL_ACCEL Acceleration when reciprocating		
Default: 16	Min. input limit: 0	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: rev./s^2	
Data type: DOUBLE	Applies from SW version:		
Meaning:	The acceleration defined here is only effective for the output of the spindle speed (MD: SPIND_OSCILL_DES_VELO) to the spindle motor. The reciprocation speed is selected using the IS “Reciprocation speed”.		
MD irrelevant for	all spindle modes other than reciprocation mode		
Related to	MD: SPIND_OSCILL_DES_VELO (reciprocation mode) IS ”Reciprocation speed” (V38032002.5) IS ”Reciprocating by PLC” (V38032002.4)		

35430 MD number	SPIND_OSCILL_START_DIR Start direction when reciprocating		
Default: 0	Min. input limit: 0	Max. input limit: 4	
Change effective after Power On	Protection level: 2/7	Unit: –	
Data type: BYTE	Applies from SW version:		
Meaning:	The spindle motor accelerates to the reciprocation speed defined in MD: Reciprocation speed (SPIND_OSCILL_DES_VELO) using the IS “Reciprocation speed“. **The start direction** is defined by MD: SPIND_OSCILL_START_DIR if the IS ”Reciprocation by PLC” is not set. MD: SPIND_OSCILL_START_DIR = 0 ——> Start direction against current direction of rotation: MD: SPIND_OSCILL_START_DIR = 3 ——> Start direction is M3 MD: SPIND_OSCILL_START_DIR = 4 ——> Start direction is M4		
MD not applicable to	all spindle modes other than reciprocation mode		
Related to	MD: SPIND_OSCILL_DES_VELO (reciprocating speed) IS ”reciprocating speed“ (V38032002.5) IS ”reciprocating by PLC” (V38032002.4)		

35440 MD number	SPIND_OSCILL_TIME_CW Reciprocation time for M3 direction		
Default: 1	Min. input limit: 0 0 corresponds to a time of **one** interpolation clock (MD: IPO_SYSCLOCK_TIME_RATIO)	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Applies from SW version:		
Meaning:	The reciprocation time is effective in the M3 direction (see illustration to MD: SPIND_OSCILL_TIME_CCW).		
MD not applicable to	• all spindle modes other than reciprocation mode • reciprocating by PLC (IS ”Reciprocating by PLC”(V38032002.4) set)		
Related to	MD: SPIND_OSCILL_TIME_CCW (reciprocation time for M4 direction) MD: IPO_SYSCLOCK_TIME_RATIO (interpolator clock) IS ”Reciprocation speed”(V38032002.5) IS ”Reciprocating by PLC” (V38032002.4)		

35450 MD number	SPIND_OSCILL_TIME_CCW Reciprocation time for M4 direction		
Default: 0,5	Min. input limit: 0 0 corresponds to a time of **one** interpolation clock (MD: IPO_SYSCLOCK_TIME_RATIO)	Max. input limit: plus	
Change effective after Power On	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Applies from SW version:		
Meaning:	The reciprocation time defined here is effective in M4 direction (see illustration).		
MD not applicable to	• all other spindle modes than reciprocation mode • reciprocating by PLC (IS ”Reciprocating by PLC” (V38032002.4) set)		

35450 MD number	SPIND_OSCILL_TIME_CCW Reciprocation time for M4 direction
	
Related to	MD: SPIND_OSCILL_TIME_CW (reciprocation time for M3 direction) MD: IPO_SYSCLOCK_TIME_RATIO (interpolator clock) IS "Reciprocation speed" (V38032002.5) IS "Reciprocating by PLC" (V38032002.4)

35500 MD number	SPIND_ON_SPEED_AT_IPO_START Feed enable for spindle in set range		
Default: 1	Min. input limit: 0	Max. input limit: 2	
Change valid after Power On	Protection level: 3/3	Unit: –	
Data type: BYTE	Valid as from SW version:		
Meaning:	**0:** The path interpolation is not influenced. **1:** The path interpolation is only enabled if the spindle has reached the specified speed (tolerance band is set via MD: SPIND_DES_VELO_TOL). **2:** Function as with value=1; in addition: Traversing axes are also stopped prior to machining start, e.g.: continuous path control mode (G64) and change from rapid traverse (G0) to a machining block (G1, G2,..). The path is stopped at the last G0 block and will only start if the spindle is in the speed setpoint range.		
Application example(s)	MD: SPIND_ON_SPEED_AT_IPO_START and this MD can be used to process the path feed as follows, depending on the spindle actual speed (control mode): D If the spindle is in the acceleration phase (programmed set speed not yet reached), the path feed is disabled. D If the actual speed deviates from the spindle speed by more than the spindle speed tolrance (MD: SPIND_DES_VELO_TOL), the path feed is enabled. D Befindet sich die Spindel in der Bremsphase, wird der Bahnvorschub gesperrt. D If the spindle is signaled as stopped (IS: "Axis/spindle stopped" " V390x0001.4), the path feed is enabled. D This control is not effective in blocks with G0.		
Related to	MD: SPIND_DES_VELO_TOL (spindle speed tolerance) NST "Spindle in set range" (V390x2001.5)		

35510 MD number	SPIND_STOPPED_AT_IPO_START Feed enable at spindle standstill		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Change valid after Power On	Protection level: 2/7	Unit: –	
Data type: BOOLEAN	Valid as from SW version:		
Meaning:	**0:** No influence on path interpolation **1:** If a spindle is stopped in control mode (M5), the path feed is only enabled if the spindle has stopped (IS, ”Axis/spindle stopped” (V390x0001.4) is set).		
Application example(s)	see MD: SPIND_ON_SPEED_AT_IPO_START		
Related to	MD: SPIND_ON_SPEED_AT_IPO_START (feed enable for spindle in setpoint range)		

Setting data

43210 SD number	SPIND_MIN_VELO_G25 Progr. spindle speed limitation G25		
Default: 0	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	User class:	Unit: Rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In SD: SPIND_MIN_VELO_G25, a min. spindle speed limitation is entered which must be reached by the spindle. The NCK limits the spindle speed to this value. The min. spindle speed can only not be reached in the following cases: ● Spindle override 0% ● M5 ● S0 ● IS ”Cancel servo enable“ (V3803002.1) ● IS ”Reset” (V30000000.7) ● IS ”Clear distance to go/Spindle Reset” (V38030002.2) ● IS ”Oscillation speed“ (V38032002.5) ● Delete S value		
SD not relevant	for any other spindle modes than control mode		
Special cases, errors, ...	The value in SD: SPIND_MIN_VELO_G25 can be modified by: ● G25 S.... in the part program ● Operation from MMC The value in SD: SPIND_MIN_VELO_G25 remains even after Reset or Power Off.		
corresponding with	SD: SPIND_MAX_VELO_G26 SD: SPIND_MAX_VELO_LIMS (progr. spindle speed limitation with G96)		

43220 SD number	SPIND_MAX_VELO_G26 Progr. spindle speed limitation G26		
Default: 1000	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	Protection level:	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In SD: SPIND_MAX_VELO_G26, a max. spindle speed limitation is entered which may not exceeded by the spindle. The NCK limits the spindle speed to this value.		
SD not applicable to	spindle modes other than control mode.		
Special cases, errors, ...	The value in SD: SPIND_MIN_VELO_G25 can be modified by : ● G25 S.... in the part program ● Operation from MMC The value in SD: SPIND_MIN_VELO_G25 remains even after Reset or Power Off.		
Related to	SD: SPIND_MIN_VELO_G25 (progr. spindle speed limitation G25) SD: SPIND_MAX_VELO_LIMS (progr. spindle speed limitation with G96)		

43230 SD number	SPIND_MAX_VELO_LIMS Progr. spindle speed limiting G96		
Default: 0	Min. input limit: 0	Max. input limit: plus	
Change valid immediately	Protection level:	Unit: rev./min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	With constant cutting rate (G96), in addition to the permanently active limits another limit is effective which is entered into this SD. In addition, this SD can be described in the part program with LIMS=....		
SD not applicable	to all spindle functions, except G96 (constant cutting rate)		
Application example(s)	When parting and in the case of very small machining diameters, as well as when constant cutting rate (G96) is selected, the spindle continues to accelerate with the workpiece (turning machine) and reaches an infinitely high set speed at the position of the transversal axis X=0. In such cases the spindle will accelerate to its max. spindle speed of the current gear stage (in some cases, limited by G26). If you wish to limit the spindle to a smaller speed, in particular, with G96, SD: SPIND_MAX_VELO_LIMS must be set.		
Special cases, errors, ...	The value in SD: SPIND_MIN_VELO_G25 can be changed by: ● G25 S.... in the part program ● Operation from MMC The value in SD: SPIND_MIN_VELO_G25 is kept even after reset oer Mains Off.		
Related to ...	SD: SPIND_MAX_VELO_G26 (max. spindle speed) SD: SPIND_MIN_VELO_G25 (min. spindle speed)		

9.7 Signal Description

Signals to axis/spindle

V38030002.2 Interface signal	Clear distance to go/ Spindle Reset Signal(s) to axis/spindle (PLC -> NCK)		
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Edge change 0 ---> 1	Irrespective of MD: SPIND_ACTIVE_AFTER_RESET, Spindle Reset is active for the individual spindle operating modes as follows: Control mode: - Spindle stops - Program is continued when G94 is active! When G95 is active, the axes are also stopped due to the missing feedrate, and the program execution is thus also stopped if G1, G2, ... are active. - Spindle runs as with G94 and a following M and S value Further: Oscillation mode: - Oscillation is aborted - Axes go on running - Program is continued with the currently active gear stage - The following M value and a higher S value might set the IS "Programmed speed too high" Positioning mode: - is stopped		
Signal status 0 or edge change 1 ---> 0	no effect		
Related to	MD: SPIND_ACTIVE_AFTER_RESET (own Spindle Reset) IS "Reset" (V30000000.7)		

V38032000.2 Interface signal	Gear stage changed Signal(s) to axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	When the new gear stage is active, the PLC user sets the IS ”Actual gear stage A to C” and the IS ”Gear stage changed”. This informs the NCK that the appropriate gear stage has been activated successfully. The gear stage change is considered completed (spindle oscillation mode is deselected), the spindle accelerates with the new gear stage to the last programmed spindle speed and the next block can be executed. The IS ”Change gear stage” is reset by the NCK, and as a response, the PLC user resets the IS ”Gear stage changed”.	
Signal status 0 or edge change 1 ---> 0	no effect	
Signal not applicable ...	to spindle modes other than oscillation mode	
Special cases, errors, ...	If the PLC user feeds back another actual gear stage than fed back as the set gear stage from the NCK to the PLC, the gear stage change is nevertheless considered completed, and actual gear stage A to C is activated.	
Related to	IS ”Actual gear stage A to C ”(V38032002.0 to .2) IS ”Set gear stage A to C” (V39032000.0 to .2) IS ”Change gear stage“ (V39032000.3) IS ”Oscillation speed“ (V38032002.5)	

V38032001.0 Interface signal	Feed override valid for spindle (instead of spindle override) Signal(s) from axis/spindle (PLC ->NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 --> 1	Instead of the spindle override value, the feed override value (VB38030000) is used.	
Signal status 0 or edge change 1 --> 0	The feed override value is used.	
Related to ...	IS ”Spindle override” (VB38032003) IS ”Feed override” (VB38030000) IS ”Override active“ (V38030001.7) see also Chapter “Feedrates”	

V38032001.6 Interface signal	Invert M3/M4 Signal(s) to axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version
Signal status 1 or edge change 0 --> 1	The direction of rotation of the spindle motor changes with the following functions: ● M3 ● M4 ● SPOS from the movement; not active with SPOS from the standstill ● Traversing the spindle in manual mode	

V38032002.0 to .2 Interface signal	Actual gear stage A to C Signal(s) to axis/spindle (PLC -> NCK)		
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 (status–controlled)	If the new gear stage is active, the PLC user sets the IS ”Actual gear stage A to C” and ”Gear stage changed”. This informs the NCK that the appropriate gear stage has been activated successfully. The gear stage change is considered completed (spindle oscillation mode is deselected), the spindle accelerates with the new gear stage to the last programmed spindle speed and the next block can be executed. The actual gear stage is specified in coded form. Each of the 5 gear stages is assigned a parameter record with the following assignment: Parameter Block No. / PLC-Interface / Data of Data Block / Contents 0 / - / Data for axis mode / Kv factor, Monitoring functions 1 / 000 / Data for 1st gear stage / M40 speed 001 / Min./max. speed ... 2 / 010 / Data for 2nd gear stage / ... 3 / 011 / Data for 3rd gear stage 4 / 100 / Data for 4th gear stage 5 / 101 / Data for 5th gear stage 110 / - 111 / -		
Special cases, errors, ...	If the PLC user feeds back another actual gear stage to the NCK than fed back as the set gear stage to the PLC, the gear stage change is nevertheless considered completed, and actual gear stage A to C is activated.		
Related to	IS ”Set gear stage A to C” (V39032000.0 to .2) IS ”Change gear stage“ (V39032000.3) IS ”Gear changed“ (38032000.3) IS ”Oscillation speed“ (V38032002.5) Parameter records for gear stages		

V38032002.7 and .6 Interface signal	Set direction of rotation CW / set direction of rotation CW Signal(s) to axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	If the IS ”Oscillation by PLC” is set, the direction of rotation for the oscillation speed can be set by both IS ”Set direction of rotation CCW and CW”. The times for the oscillation motion of the spindle motor are defined by setting the IS ”Set direction of rotation CCW and CW” accordingly long.	
Signal not applicable ...	to spindle modes other than oscillation	
Application example(s)	see IS ”Oscillation by PLC”	
Special cases, errors, ...	● If both IS are set at the same time, no oscillation speed is output. ● If no IS is set, no oscillation speed is output.	
Related to	IS ”Oscillation by the PLC” (V38032002.4) IS ”Oscillation speed“ (V38032002.5)	

V38032002.5 Interface signal	Oscillation speed Signal(s) to axis/spindle (PLC -> NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	If you wish to carry out gear stage change (IS "Change gear stage" (V39032000.3) is set), the spindle mode changes to oscillation mode. Depending on at which moment the IS "Oscillation speed" (V38032002.5) is set, the spindle decelerates to standstill with different accelerations: 1. The IS "Oscillation speed" is set **before** the IS "Change gear stage" is set by the NCK. The spindle is decelerated to standstill with the acceleration of oscillation (MD: SPIND_OSCILL_ACCEL). If the spindle is on standstill, oscillation is started immediately. 2. The IS "Oscillation speed" is set **after** the IS "Change gear stage" has been set by the NCK and after the spindle has come to stop. The position control is switched off. The spindle is decelerated with the acceleration of speed–controlled mode. When the IS "Oscillation speed" is set, the spindle starts oscillating with the oscillation acceleration (MD: SPIND_OSCILL_ACCEL). If the IS "Oscillation by PLC" (V38032002.4) **is not set**, automatic oscillation in the NCK is carried out using the IS "Oscillation speed". The two times for the directions of rotation are entered in the MD: SPIND_OSCILL_TIME_CW (oscillation time for M3 direction) and MD: SPIND_OSCILL_TIME_CCW (oscillation time for M4 direction). If the IS "Oscillation by PLC" **is set**, the IS "Oscillation speed" will only output a speed in conjunction with the IS "Set direction of rotation CW and CCW". Oscillation, i.e. permanent changing of the direction of rotation, is carried out by the PLC user with the IS "Set direction of rotation CCW and CW" (oscillation by PLC).	
Signal status 0 or edge change 1 ---> 0	The spindle does not oscillate.	
Signal not applicable	to spindle operating modes other than oscillation mode	
Application example(s)	Oscillation speed is used to facilitate meshing of the new gear stage. The spindle motor must permanently change its direction of rotation.	
Related to	IS Oscillation by the PLC (V38032002.4) IS Set direction of rotation CCW (V38032002.7) IS Set direction of rotation CW (V38032002.6)	

V38032002.4 Interface signal	Oscillation by the PLC Signal(s) to axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	If the IS "Oscillation by the PLC" **is not set**, automatic oscillation in the NCK is carried with IS "Oscillation mode". The two times for the directions of rotation are entered in the MD: SPIND_OSCILL_TIME_CW (oscillation time for M3 direction) and MD: SPIND_OSCILL_TIME_CCW (oscillation time for M4 direction). If the IS "Oscillation by the PLC" **is set**, a speed with the IS "Oscillation speed" is only output in conjunction with the IS "Set direction of rotation CW and CCW". Oscillation, i.e. permanent change of direction, is carried out by the PLC user by means of the IS "Set direction of rotation CCW and CW" (oscillation by the PLC).	
Application example(s)	If the gear stage cannot be activated during oscillation by the NCK instead of several tries, it is possible to change to oscillation by the PLC. When this possibility is used, the two times for the directions of rotation can be modified by the PLC user as he wants. This ensures that safe changing of the gear stage is possible even in case of bad toothed gear positions.	
Related to	MD: SPIND_OSCILL_TIME_CW (oscillation time for M3 direction) MD: SPIND_OSCILL_TIME_CCW (oscillation time for M4 direction) IS "Oscillation speed" (V38032002.5) IS "Set direction of rotation CCW" (V38032002.7) IS "Set direction of rotation CW" (V38032002.6)	

VB38032003 Interface signal	Spindle override Signal(s) to spindle (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 --> 1	The spindle override is specified from the spindle gray–coded. The override value determines the percentage of the programmed speed setpoint output to the spindle.	

Switch Position	Code	Spindle Override Factor
1	00001	0.5
2	00011	0.55
3	00010	0.60
4	00110	0.65
5	00111	0.70
6	00101	0.75
7	00100	0.80
8	01100	0.85
9	01101	0.90
10	01111	0.95
11	01110	1.00
12	01010	1.05
13	01011	1.10
14	01001	1.15
15	01000	1.20
16	11000	1.20
17	11001	1.20
18	11011	1.20
19	11010	1.20
20	11110	1.20
21	11111	1.20
22	11101	1.20
23	11100	1.20
24	10100	1.20
25	10101	1.20
26	10111	1.20
27	10110	1.20
28	10010	1.20
29	10011	1.20
30	10001	1.20
31	10000	1.20

Table 9-1 Gray coding for spindle override

Related to ...	IS "Override active" (V38030001.7) IS "Feed override valid for spindle" (V38032001.0)

Signals from axis/spindle

V39030000.0 Interface signal	Spindle – no axis Signal(s) from axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The machine axis is operated **as a spindle i**n the following spindle modes: • Control mode • Oscillation mode • Positioning mode The IS to axis (VB38031000 to ... 03) and from axis (VB39031000 to ... 03) are invalid. The IS to spindle (VB38032000 to ... 03) and from spindle (VB39032000 to ... 03) are valid.	
Signal status 0 or edge change 1 ---> 0	The machine axis is operated **as an axis.** The IS to axis (VB38031000 to ... 03) and from axis (VB39031000 to ... 03) are valid. The IS to spindle (VB38032000 to ... 03) and from spindle (VB39032000 to ... 03) are invalid.	
Application example(s)	Spindle override	

V39032000.3 Interface signal	Change gear stage Signal(s) from axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	A gear stage can be set: ● fixed by the part program (M41 to M45) ● automatically by the programmed spindle speed (M40) M41 to M45: ● The gear stage can be set in the part program by means of M41 to M45 as a fixed setting. If as gear stage is set by M41 to M45 which is other than the current (actual) gear stage, the IS “Change gear stage” and the IS “Set gear stage A to C” are set. M40: ● The control system uses M40 is used in the part program to set the gear stage automatically. A check is carried out to see in which gear stage the programmed spindle speed is possible (S function). If a gear stage other than the current (actual) gear stage is found, the IS “Change gear stage” and the IS “Set gear stage A to C” are set. ● When the signal = 1, the text “Waiting for gear stage change” is displayed in the channel operating message line.	
Special cases, errors, ...	The IS ”Change gear stage” is only set if the new gear stage is **other** than the currently active gear stage.	
Related to	MD: GEAR_STEP_USED_IN_AX_MODE (gear stage for rotary axis mode) IS ”Set gear stage A to C” (V39032000.0 to .2) IS ”Actual gear stage A to C” (V38032000.0 to .2) IS ”Gear stage changed“ (V38032000.3)	

V39032000.0 to .2 Interface signal	Set gear stage A to C Signal(s) from axis/spindle (PLC -> NCK)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	A gear stage can be set: ● fixed by the part program (M41 to M45) ● automatically by the programmed spindle speed (M40) M41 to M45: ● The gear stage can be set in the part program by means of M41 to M45 as a fixed setting. If as gear stage is set by M41 to M45 which is other than the current (actual) gear stage, the IS “Change gear stage” and the IS “Set gear stage A to C” are set. M40: ● The control system uses M40 is used in the part program to set the gear stage automatically. A check is carried out to see in which gear stage the programmed spindle speed is possible (S function). If a gear stage other than the current (actual) gear stage is found, the IS “Change gear stage” and the IS “Set gear stage A to C” are set. The set gear stage is output in coded form: 1st gear stage 0 0 0 (C B A) 1st gear stage 0 0 1 2nd gear stage 0 1 0 3rd gear stage 0 1 1 4th gear stage 1 0 0 5th gear stage 1 0 1 invalid value 1 1 0 invalid value 1 1 1	
Signal not applicable	to spindle modes other than oscillation	
Related to	IS ”Change gear stage“ (V39032000.3) IS ”Actual gear stage A to C” (V38032000.0 to .2) IS ”Gear stage changed” (V38032000.3)	

V39032001.7 Interface signal	Actual direction of rotation CW Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	When the spindle rotates, CW direction of rotation is signalled by the IS "Actual direction of rotation CW" = 1. The actual direction of rotation is derived from the spindle actual position encoder.	
Signal status 0 or edge change 1 ---> 0	When the spindle rotates, the CCW direction of rotation is signalled by the IS "Actual direction of rotation CCW" = 0:	
Signal not applicable	• Spindle at a standstill, IS "Axis/spindle on standstill" = 1 (at a standstill, no evaluation of the direction of rotation possible) • Spindles without position encoder	
Related to	IS "Spindle on standstill"	

V39032001.5 Interface signal	Spindle within set range Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The IS "Spindler within set range" signals whether the programmed and/or limited (if so) spindle speed is reached. In spindle mode "Control mode", the set speed (programmed speed * spindle override with consideration of the limits) is compared with the actual speed. If the actual speed deviates from the set spindle speed by **less** than the spindle speed tolerance (MD: SPIND_DES_VELO_TOL), the IS "Spindle within set range" is set.	
Signal status 0 or edge change 1 ---> 0	The IS "Spindle within set range" signals whether the spindle is still in the acceleration phase. In spindle mode "Control mode", the set speed (programmed speed * spindle override with consideration of the limits) is compared with the actual speed. If the actual speed deviates from the set speed by **more** than the spindle speed tolerance (MD: SPIND_DES_VELO_TOL) from the set speed, the IS "Spindler within set range" is reset.	
Signal not applicable	to spindle modes other than speed–controlled mode (control mode)	
Related to	MD: SPIND_DES_VELO_TOL (spindle speed tolerance)	

V39032001.1 Interface signal	Set speed limited Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	If a spindle speed (1/min) or a constant cutting speed (m/min or ft/min) has been programmed, one of the following limit values has been exceeded: • Max. speed of the set gear stage • Max. spindle speed • Speed limitation by PLC interface • Progr. spindle speed limitation G26 • Progr. spindle speed limitation with G96 The spindle speed is limited to the max. limit value.	
Signal status 0 or edge change 1 ---> 0	If a spindle speed (1/min) or constant cutting speed (m/min or ft/min) has been programmed, no limit values have been exceeded.	
Application example(s)	The IS "Set speed limited" can be used to detect that the programmed speed cannot be reached. The PLC user can either recognize this status as admissible and enable the path feed or he can block the path feed or the entire channel, and IS "Spindle within set range" is processed.	

V39032001.2 Interface signal	Set speed increased Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	If a spindle speed (1/min) or a constant cutting speed (m/min or ft/min) was programmed, one of the following limit values has not been reached: • Min. speed of the set gear stage • Min. spindle speed • Progr. spindle speed limitation G25 The spindle speed is limited to the min. limit value (increased).	
Signal status 0 or edge change 1 ---> 0	If a spindle speed (1/min) or constant cutting speed (m/min or ft/min) was programmed, no limit values have been exceeded.	
Application example(s)	The IS ”Set speed increased” can be used to detect that the programmed speed cannot be reached.	

V39032001.0 Interface signal	Speed limit exceeded Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	If the actual speed exceeds the max. spindle speed (MD: SPIND_MAX_VELO_ LIMIT) by more than the spindle speed tolerance (MD: SPIND_DES_VELO_TOL), the IS ”Speed limit exceeded” is set and alarm 22050 output. All axes and spindles of the channel are decelerated.	
Related to	MD: SPIND_DES_VELO_TOL (spindle speed tolerance) MD: SPIND_MAX_VELO_LIMIT (max. spindle speed) Alarm 22050 ”Max. speed reached”	

V39032002.7 Interface signal	Active spindle mode: Control mode Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	When the following functions are active, the spindle is in control mode: • Setting of direction of rotation of spindle M3/M4 or Spindle Stop M5 • M41...M45, or automatic gear stage change	
Related to	IS ”Spindle mode Oscillation Mode“ (V39032002.6) IS ”Spindle mode Positioning Mode” (V39032002.5)	

V39032002.6 Interface signal	Active spindle mode: Oscillation mode Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The spindle is in oscillation mode if a new gear stage has been set either by automatic gear stage selection (M40) or by M41 to M45 (IS ”Change gear stage” is set). The IS ”Change gear stage” is only set if the new gear stage is **other** than the currently active gear stage.	
Related to	IS ”Spindle mode Control Mode“ (V39032002.7) IS ”Spindle mode Oscillation Mode” (V39032002.5) IS ”Change gear stage” (V39032000.3)	

V39032002.5 Interface signal	Active spindle mode: Positioning mode Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	When the following function is active, the spindle is in positioning mode: SPOS=	
Related to	IS ”Spindle mode Control Mode” (V39032002.7) IS ”Spindle mode Oscillation Mode” (V39032002.6)	

V39032002.3 Interface signal	Tapping without compensating chuck active Signal(s) from axis/spindle (NCK -> PLC)	
Edge evaluation: yes	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The tapping function G331, G332 is active. No response has been provided, or the spindle–specific signals, such as: IS “Spindle Reset” IS “Invert M3/M4” IS “Spindle within set range” IS “Set speed increased” have not been updated. Note: During tapping without compensating chuck, the following functions should not be used: Cancel IS “Servo enable” Set IS “Feed Stop” Set IS “Reset”	

Output of Auxiliary Functions to PLC

10

Brief description

For machining workpieces on a machine tool, in addition to axis positions and interpolation types, the CNC can also set technological functions (feed, spindle speed, gear stage, tool change) and functions to control additional equipment on the machine tool (e.g., sleeve, open gripper, clamp chuck, etc.).

The following auxiliary functions can be output to the PLC:

- **Miscellaneous function M**
- **Tool number T**

These functions come into effect at defined moments during program execution and are output to the PLC.

Functions/block

Per part program, the following functions can be programmed:

- five M functions
- one S function
- one T function
- one D function
- one F function

In one block, max. 10 auxiliary funtions can be programmed,
e.g. N10 S3000 T1 D2 M3 M77 M87 ...

If the permissible number of auxiliary functions per block is exceeded, alarm 12010 is output.

Block change

Only after the PLC operating system has acknowledged all transferred auxiliary functions, the NCK can output a new auxiliary function to the PLC.
A block is considered completed if the programmed movement is completed and the auxiliary function is acknowledged. To this aim, the part program execution is stopped by the NCK in order to ensure that no auxiliary functions get lost, from the viewpoint of the PLC user program.

Continuous–path control mode

A path motion will only remain continuously if the auxiliary function is output during the motion and acknowledged before the path end.

10.1 Auxiliary Function Groups

Functionality

The auxiliary functions of the auxiliary function groups M and T can be divided by means of machine data into auxiliary function groups.

An auxiliary function may only be assigned to one group.

Only one auxiliary function per block may be programmed. Otherwise, alarm 14760 is output.

Configuration

A maximum of 15 auxiliary function groups can be defined.
These 15 auxiliary function groups can be assigned max. 50 auxiliary functions (per channel).
The pre–assigned auxiliary functions are not yet considered in this number.
The real number of auxiliary functions distributed over the groups is entered in the NCK–specific MD: AUXFU_MAXNUM_GROUP_ASSIGN (number of auxiliary functions distributed over the auxiliary function groups).

Assigned auxiliary functions are defined in the following machine data:

AUXFU_ASSIGN_TYPE[n]	auxiliary function type
AUXFU_ASSIGN_VALUE[n]	auxiliary function value
AUXFU_ASSIGN_GROUP[n]	auxiliary function group

Pre–assigned auxiliary function groups

The pre–assigned auxiliary function groups have the following behavior:

- Output at block end (group 1)
- Output prior to the movement (group 2)

Group 1:
By default, the auxiliary functions M0, M1 and M2 are assigned to group 1.

Group 2:
By default, the M functions M3, M4 and M5 are assigned to group 2.

10.2 Behavior with Block Search

Block search with calculation

During block search with calculation, all auxiliary functions assigned to a group are accumulated and output at the end of the block prior to the block at which program execution is continued (except for group 1: M0, M1,...).
Always the last auxiliary function of a group is output.

All accumulated auxiliary functions are output prior to the movement in a separate block as normal auxiliary functions.

Important: If all auxiliary functions are to be accumulated during block serach, they must be assigned to one and the same auxiliary function group!

10.3 Description of Auxiliary Functions

M function

Application

The M functions can be used to activate various switching actions on the machine via the part program.

Scope of functions

- 5 M functions per part program block are possible.
- Value range of M functions: 0 to 99; integer
- A minor part of the M functions is assigned by the control manufacturer a fixed functionality (see User Manual “Operation and Programming”). The remaining part is available to the machine manufacturer for free use.

T function

Application

The T function is used to load the appropriate tool for a certain machining section from the PLC. Whether the tool change is to be carried out directly by means of the T command or a following M command can be set via machine data (see User Manual “Operation and Programming“).

The programmed T function can be interpreted either as a tool No. or as a tool magazine position No.

Scope of functions

1 T function per part program block is possible.

Special property

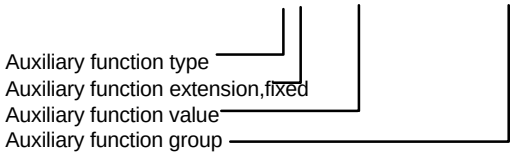
T0 is reserved for the following function: to remove the currently active tool from the tool holder without loading a new tool.

10.4 Data Description

Machine data

11100 MD number	AUXFU_MAXNUM_GROUP_ASSIGN Number of auxiliary functions distributed over the auxiliary function groups		
Default: 1	Min. input limit: 1	Max. input limit: 50	
Changes effective after Power On	User class: 2/7	Unit: -	
Data type: BYTE	Valid as from SW version:		
Meaning:	In this MD, the real number of auxiliary functions which have been distributed over the groups must be entered. Only the customer–specific machine data are relevant, not the predefined auxiliary functions.		
Application example(s)			
Related to	MD 22010: AUXFU_ASSIGN_TYPE[n]		

22000 MD number	AUXFU_ASSIGN_GROUP[n] Auxiliary function group [HiFunr.]: 0...49		
Default: 1	Min. input limit: 1	Max. input limit: 15	
Changes effective after Power On	User class: 2/7	Unit: -	
Data type: BYTE	Valid as from SW version:		
Meaning:	see MD: AUXFU_ASSIGN_TYPE [n] (auxiliary fucntion type)		
Application example(s)			

22010 MD number	AUXFU_ASSIGN_TYPE[n] Auxiliary function type [HiFunr. in channel]: 0...49		
Default: -	Min. input limit: -	Max. input limit: 16 characters	
Changes effective after Power On	User class: 2/7	Unit: -	
Data type: STRING	Valid as from SW version:		
Meaning:	The machine data AUXFU_ASSIGN_TYPE[n] (auxiliary function type), AUXFU_ASSIGN_VALUE[n] (auxiliary function value) and AUXFU_ASSIGN_GROUP[n] (auxiliary function group) are used to assign an auxiliary function type (M,T), the associated extension and the auxiliary function value to an auxiliary function group. Example: M 0 = 99 => group 5 (corrsp. to M99)  ⇒ MD: AUXFU_ASSIGN_TYPE[0] = ”M” MD: AUXFU_ASSIGN_VALUE[0] = 99 MD: AUXFU_ASSIGN_GROUP[0] = 5; (≒ 5th group) By default, M0, M1, M2, (M17 and M30) are assigned to group 1; by default, M3, M4, M5 are assigned to group 2. The index [n] of the machine data denotes the auxiliary function number: 0-49 All auxiliary functions that are assigned to auxiliary function groups must be numbered in ascending order. [0] ≒ 1st auxiliary function [1] ≒ 2nd ,, . . The three machine data used to assign an auxiliary function to an auxiliary function group must be assigned the same index [n].		
Application example(s)	see Chapter 6		
Special cases, errors, ...	If the auxiliary function value of an auxiliary function is less than 0, all auxiliary functions of this type and extension are assigned to one group.		
Related to	MD 11100: AUXFU_MAXNUM_GROUP_ASSIGN		

22030 MD number	AUXFU_ASSIGN_VALUE[n] Auxiliary function value [HiFuNr.]: 0...49		
Default: 0	Min. input limit: ***	Max. input limit: ***	
Changes effective after Power On	User class: 2/7	Unit: -	
Data type: DWORD	Valid as from SW version:		
Meaning:	see MD: AUXFU_ASSIGN_TYPE[n] (auxiliary function type)		
Application example(s)	see Chapter 6		

10.5 Signal Description

V25000000.0 and V25000001.4 Interface signal	M function change T function change Signal(s) from channel (NCK ---> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge change 0 ---> 1	An M/T information with a new value has been output to the interface, in conjunction with the associated change signal. The change signals indicate that the respective value is valid.		
Signal status 0 or edge change 1 ---> 0	The change signals are reset at the beginning of the next cycle by the PLC system program. The value of the respective information is not valid.		

VD25002000 Interface signal	T function 1 Signal(s) from channel (NCK ---> PLC)		
Edge evaluation: no	Signal(e) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge evaluation 0 ---> 1	This data makes available the T function programmed in an NC block, as soon as the T change signal is present. Value range of T function: 0-99 ; integer The T function remains until it is overwritten by a new T function.		
Signal status 0 or edge change 1 ---> 0	• After power–up of PLC • Before a new auxiliary function is entered, the other auxiliary functions are deleted.		
Application example(s)	Control of automatic tool selection		
Special cases, errors, ...	T0 is intended to remove the currently active tool from the tool holder; no new tool is loaded (default configuration of machine manufacturer).		

VB25001000 to Interface signal	Dynamic M functions: M0 - M99 Signal(s) from channel (NCK ---> PLC)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge evaluation 0 ---> 1	The dynamic M signal bits are set by decoded M functions.		
Signal status 0 or edge evaluation 1 ---> 0	During general auxiliary function output, the dynamic M signal bits are acknowledged by the PLC system program after the user program has been completely executed once.		
Application example(s)	CW/CCW rotation of spindle, Coolant ON/OFF		

Feeds

11

11.1 Overview

Feed types

The feedrate is the machining speed with which the tools moves along the programmed workpiece contour (path). The individual axis speeds according to their portion in the path result from this path.
Workpiece machining additionally requires a rotating spindle. The spindle speed is set separately, e.g. via the program with address S.

Apart from different machining tasks, various positioning processes,too, are possible. In this case, the workpiece traverses along a straight line, but not on the workpiece at the highest possible traversing rate.

Depending on the activated interpolation type and special G commands for feed preselection, different feedrates/velocities are used in the program. The same applies to program dry run or manual traversing:

- Feed F with G1, G2, G3, G5
- Feed for thread cutting G33
- Feed for tapping with compensating chuck G63
- Feed for tapping without compensating chuck G331, G332
- Rapid traverse with G0
- Dry run feed
- Velocities for manual traversing of the axes

Feed override

For adaptation to modified technological conditions during machining or for testing purposes, the programmed feed can be modified via operation/PLC, e.g. by turning the feed override switch or activating dry run feed.

11.2 Feed F

Functionality

The feed F is the **path velocity** of the tool along the programmed workpiece contour.
The individual axis velocities result from the portion of the axis path in the contour path.
The feed F is active for the interpolation types G1, G2, G3, G5 and remains active in the program until a new F word is written.
(see User Manual “Operation and Programming“)

Programming

F...

Note**:** For **integer values**, the decimal point need not be specified, e.g. F300.

Unit for F-G94, G95

The unit of the F word is defined by G functions:

- G94 Feed **F** in **mm/min**
- G95 Feed **F** in **mm/rev.** of the spindle
  (makes only sense if the spindle rotates!)

Programming example

```
N10 G94 F310  ;Feed in mm/min
…
N110 S200 M3  ;Spindle rotation
N120 G95 F15.5                ;Feed in mm/rev.
```

Unit for F with G96, G97

For **turning machines,** the group with G94, G95 is extended by the functions G96, G97 for **constant cutting speed** (ON/OFF). These functions have an additional influence on the S word.
When the function G96 is active, the spindle speed is adapted to the diameter of the workpiece currently being machined (face axis) such that the programmed cutting speed S on the tool edge remains constant (spindle speed multiplied with diameter = constant).
From the block containing G96, the S word is interpreted as the cutting speed. G96 is modally active until it is canceled by another G function of the group (G94, G95, G97).
The **feed F** is here always interpreted in the unit **mm/rev.** (as with G95).

Note

When a G command that requires a new unit for the F word is changed, the F value must be modified, too.

Maximum path velocity

The max. path velocity results from the maximum velocities of the axes involved (MD: MAX_AX_VELO) and their portion in the contour path. The maximum axis velocity stored in the MD cannot be exceeded.

Feed override for circles G901

When circle contours are machined by milling tools and with tool radius compensation (G41/G42) switched on, the feed on the cutter center point must be corrected if the programmed F value is to act on the circle contour. With feed override (G901) active, internal and external circle machining are detected automatically.
The feed override can be switched off by G900.

Interface signal

When revolutional feed is active, the IS “Revolutional feed active” (V33000001.2) is set.

Alarms

- If no F word for G1, G2, G3, G5 is programmed, alarm 10860 is output. No axis movement can be carried out.
- When F0 is programmed, alarm 14800 is output.
- If the spindle is on standstill when G95 is active, no axis movement can be carried out. No alarm is output.

11.3 Feed for Thread Cutting G33

Application

The function G33 can be used to machine threads with constant lead. This function can also be used for tapping with compensating chuck.
Detail description: see User Manual “Operation and Programming“

Axis velocity

For G33 threads, the axis velocity for the thread length results from the set spindle speed and the programmed thread lead. However, the maximum axis velocity defined by MD: MAX_AX_VELO cannot be exceeded.
The **feed F is not relevant** but remains stored.

The axis velocity for the thread length is calculated from the set spindle speed (S) and the programmed thread lead of this axis.

e.g., for a cylinder thread:

F_Z [mm/min] = speed S [rev./min] * thread lead K[mm/rev.]

NC STOP, Single Block

NC STOP and Single Block are only active at the end of a thread chain.

Information

Important

- The spindle speed override switch should remain unchanged when the thread is machined.
- The feed override switch is not relevant for blocks with G33.

11.4 Feed for Tapping with Compensating Chuck G63

Application

G63 is a partial function for tapping with tap in a compensating chuck. A position measuring system on the spindle is not required.
Detail description: see User Manual “Operation and Programming”

Feed F

A new feed F for G63 must be programmed. It must match with the selected spindle speed S (programmed or set) and the thread lead of the tap:

Feed F[mm/min] = speed S [rev./min] x thread lead [mm/rev.]

The compensating chuck compensates any path differences of the tap axis to a certain degree.

11.5 Feed for Tapping without Compensating Chuck G331, G332

Application

G331 – tapping – and G332 – retraction from tapping – can be used for tapping without compensating chuck. However, the spindle must have the technological prerequisites to change to position control mode. For this reason, spindle positioning with SPOS=... must be programmed before tapping.

Detail description: see User Manual “Operation and Programming“

Axis velocity

For tapping with G331/G332, the axis velocity for the thread length results from the programmed spindle speed S and the programmed thread lead. However, the maximum axis velocity defined by MD: MAX_AX_VELO cannot be exceeded.
The **feed F does not apply** but remains stored.

11.6 Rapid Traverse G0

Rapid traverse motion G0 is used for high–speed positioning of the tool but **not for direct workpiece machining.**
All axes can be traversed at the same time. This results in a straight path.

The maximum velocity (rapid traverse) for each axis is defined in machine data (MD:MAX_AX_VELO). When only one axis traverses, it traverses at rapid traverse rate. When two axes are traversed at the same time, the path velocity (resulting velocity) is selected such that both axes are traversed with the **maximum possible path velocity**.
For example, if both axes are to be traversed with the same maximum velocity and have to cover the same path distance, the

path velocity = 1.41 * max. axis velocity

(geometrical total of the two axis components).

The feed F is not relevant for G0 but remains stored.

Rapid traverse override

Via the operation -> Program Control softkey, it is possible to program the feed override switch such that it is also active for rapid traverse. When the function is active, “ROV” is displayed in the status line. The IS “Feed override for rapid traverse selected” (V17000001.3) is set from the PLC to the MMC.
More detail description: see Section 11.9.2 “Feed override via Machine Control Panel”.

11.7 Dry Run Feed

Functionality

This function is used for testing programs. When the Dry Run function is active and the program started, the feedrates programmed in conjunction with G1, G2, G3, G5 are replaced by the feed value programmed in SD: DRY_RUN_FEED. The dry run feed value is also valid instead of the programmed revolutional feed in program blocks with G95.
However, if the programmed feedrate is higher than the dry run feedrate, the higher value is used.

Danger

When the Dry Run Feed function is active, workpiece machining is not allowed, since the cutting speed of the tools might be exceeded and the workpiece or machine tool could be destroyed due to the modified feedrate values.

Selection

The operation with dry run feed is selected in the operator interface in the Program Control menu. This sets the IS “Dry run feed selected” (V17000000.6) to the PLC.
In addition, the desired value for the dry run feed must be entered in the Setting Data menu.
With the function activated, “DRY” is displayed on the status display.
(see also Section 5.2 Program Test)

Dry run feed change

The dry run feed in SD: DRY_RUN_FEED should be modified prior to program start (NC Start) -> operation: softkey “Parameter/Setting Data“.
Any changes after program start have no effect.

Activation

The interface signal ”Activate dry run“ is evaluated with NC Start if the channel was in Reset status.

11.8 Velocity for Manual Traversing

JOG mode

For traversing the axes by manual operation (in the following called 'manual traversing'), JOG mode must be active.
JOG mode differentiates various JOG variants (the so–called machine functions):

- continuous traversing (as long as the traversing key for the axis is pressed)
- incremental traversing (preselected number of increments)

Simultaneous traversing

In JOG mode, all axes can be traversed at the same time.
If several axes are traversed simultaneously, they are not involved in interpolation.

Velocity

The velocity of a traversing movement of an axis in JOG mode is defined by a setting data:
SD: JOG_SET_VELO (JOG velocity in mm/min)
The value can be entered via operation: Softkey “Parameter” -> “Setting Data”.
If the value is zero, the axis will traverse with the value of the axis machine data MD:JOG_VELO.

Rapid traverse override

When the rapid traverse override key is pressed together with the traversing keys, the movement is carried out at the rapid traverse rate defined by the axis–specific MD:
JOG_VELO_RAPID (axis velocity in JOG mode with rapid traverse override).

Feed override

The velocity used for traversing in JOG mode can additionally be controlled by means of the axis feed override switch.

(For more detailed information on manual traversing in JOG mode, see Section “Manual Traversing and Handwheel Traversing“)

11.9 Feed Override

Feed programming and feed override

The illustration below shows the possibilities of feed programming and feed control (override).

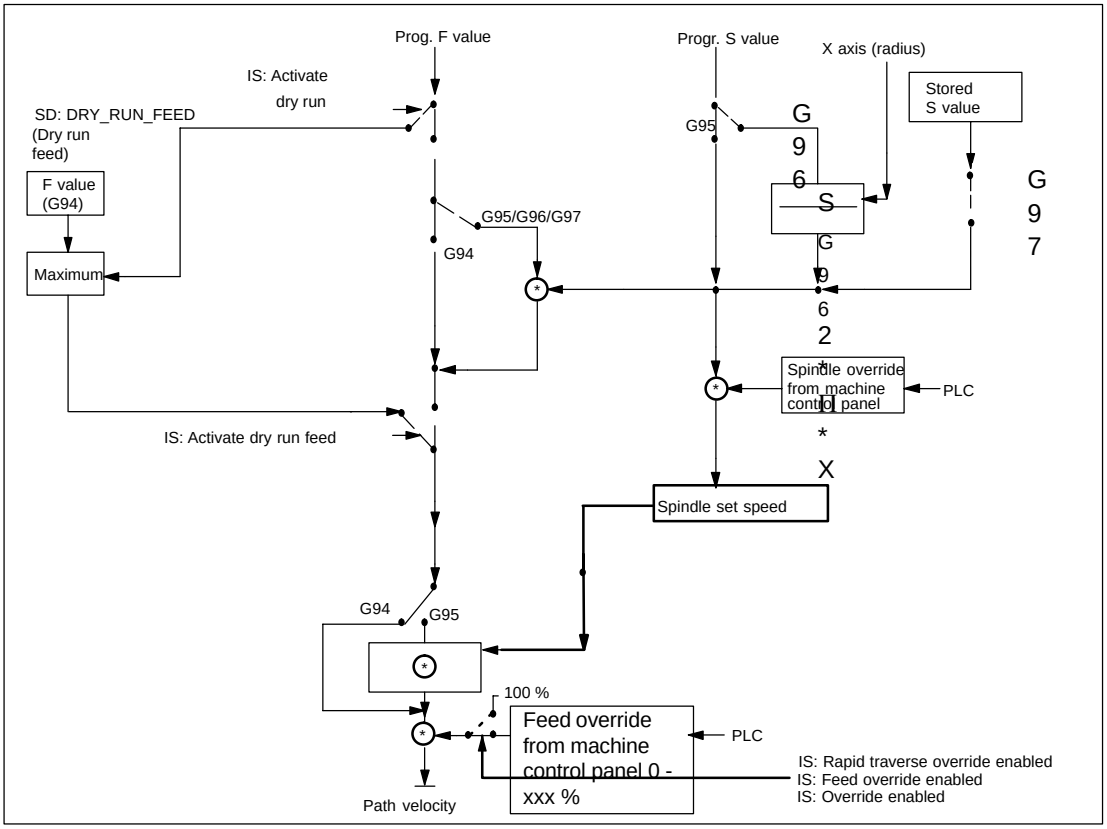


Fig. 11-1 Feed programming and feed override

11.9.1 Feed Lock and Feed/Spindle Stop

General

In case of Feed Lock or Feed/Spindle Stop, the axes are brought to stop. The path contour is not violated (exception: G33 block).

Feed Lock

The interface signal ”Feed lock“ (V32000006.0) sets all axes to standstill in all operating modes.

Channel–specific feed blocking is **not active** when **G33** is active (only for G63, G331, G332).

Feed Stop for axes in WCS

The interface signal ”Feed Stop“ (V32001000.3 and V32001008.3) sets the respective axes traversed in the workpiece coordinate system (WCS) in JOG mode to standstill.

Axis–specific Feed Stop

The axis–specific interface signal ”Feed Stop“ (V380x0004.3) sets the respective machine axis to standstill.

The following applies to Automatic mode:

- If “Feed Stop” for one contour axis is provided, all axes moved in the current block and involved in the contour complex are set to standstill.

In JOG mode, only the axis concerned is set to standstill.

Axis–specific Feed Stop is active when G33 is active
(but: deviations from contour = thread errors occur!).

Spindle Stop

The interface signal ”Spindle Stop“ (V38030004.3) sets the respective spindle to standstill.

“Spindle Stop” is active when G33 is active
(but: deviations from contour = thread errors occur!).

11.9.2 Feed Override from Machine Control Panel

General

The operator can use thefeed override switch to reduce or increase the traversing path feedrate relatively to the programmed feedrate in per cent right on the spot and with immediate effect. The feedrates are multiplied with the override values.

The override possible for the path feed F

ranges from 0 to 120%.

Therapid traverse override switch is used to slow down the traversing process when part programs are tested.

The override possible for the rapid traverse ranges from 0 to 100%.

The spindle override can be used to modify the spindle speed and the cutting speed (with G96). An override between 50 and 120% is possible.

A modification is carried out with consideration of the machine–specific acceleration and velocity limits without contour violation.

The overrides are active for the **programmed values** before the limits (e.g. G26) are active.

Channel–specific feed and rapid traverse override

For feed and rapid traverse, one enable signal each and one byte for the override factor in per cent are provided by the PLC interface.

IS ”Feed override“ (VB32000004)

IS ”Feed override active” (V32000006.7)

IS ”Rapid traverse override“ (VB32000005)

IS ”Rapid traverse override active” (V32000006.6)

The interface for the override (value) is entered from the machine control panel via the PLC in the form of a Gray code.

An active feed override applies to for all contour axes.
An active rapid traverse override is valid for all axes traversing at rapid traverse rate.

If **no own rapid traverse override switch exists,** the feed override switch can be used; in this case, feed overrides of more than 100% are limited to 100% rapid traverse override.

Which override is to be active can be selected either via the PLC or from the operator panel. When the override is selected via the operator panel (display: ROV), the IS
“Feed override for rapid traverse selected” (V17000001.3) is set, transferred to the IS “Rapid traverse override active” (V32000006.6) and the value of the
IS “Feed override“ (VB1000004) from the machine control panel (MCP) is also transferred to the IS “Rapid traverse override“ (VB3200005).

The channel–specific feed and rapid traverse override are inactive when G33, G63 G331 and G332 are active.

Axis–specific feed override

One enable signal and one byte each for the feed override factor in per cent are provided for each axis in the PLC interface.

IS ”Feed override“ (VB380x0000)
IS ”Override active“ (V380x0000.7)

The axis–specific feed override is inactive when G33 is active.

Spindle override

One enable signal and one byte each for the spindle override factor in per cent are provided by the PLC interface.
IS ”Spindle override“ (VB38032003)
IS ”Override active“ (V3803000.7)
Another signal is provided
(IS ”Feed override for spindle active” (V38032001.0)
which can be used by the PLC user program to set that the value of the
IS “Feed override“ (VB38030000) is to be active.

The value from the machine control panel (MCP) is available in the
IS “Spindle override“ (VB10000005).

Spindle override is active when G33 is active - however, for accuracy reasons it should not be actuated.

Note:
A separate spindle override switch on the machine control panel (MCP) is provided as an option.

Override active

The override values set via selector switches on the machine control panel are immediately active in all operating modes and machine functions provided the IS ”Rapid traverse override active”, ”Feed override active” or ”Override active” are set.

An override value of 0% acts as a feed lock.

Override inactive

When the override is inactive (above mentioned IS signals set to ”0”), the NC internally uses override factor ”1”, i.e. the override amounts to 100%. The value entered in the PLC interface is not relevant.

Spindle override reference

The spindle override controls the programmed speed.

11.10 Data Description

Setting data

42100 SD number	DRY_RUN_FEED Dry run feed		
Default: 5000	Min. input limit: 0	Max. input limit: plus	
Changes effective immediately	User class:	Unit: mm/min	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In this setting data, the feed for the active dry run is entered. The setting data can be modified via the operator panel in the operating area ”Parameter”. The entered dry run feed is always interpreted as the linear feed (G94). If dry run feed is activated via the PLC interface, not the programmed but the dry run feed is used after RESET. If the programmed speed is higher than the stored speed, the programmed speed is used for traversing.		
Application example(s)	Testing of programs		
Related to	IS ”Activate dry run feed“, IS ”Dry run feed selected“		

11.11 Signal Descriptions

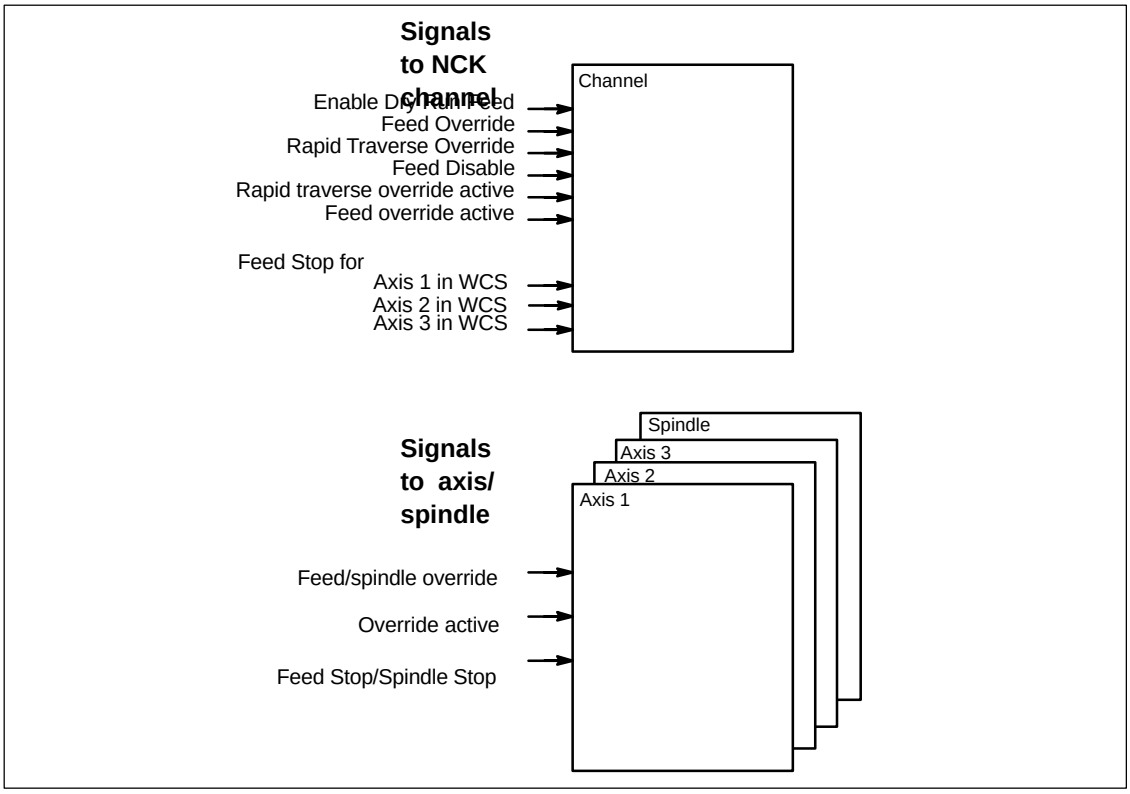


Fig. 11-2 PLC interface signals for feeds

11.11.1 Signals to Channel

V32000000.6 Interface signal	Activate dry run feed Signal(s) to channel (PLC → NCK)		
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:	
Signal status 1 or edge evaluation 0 ---> 1	If the dry run feed is higher than the programmed feed, the dry run feed defined by SD: DRY_RUN_FEED is used for traversing instead of the programmed feedrate (with G1, G2, G3, G5). Dry run feed is active after Reset status. The interface signal is evaluated when NC Start is received and the channel has been in the Reset status. Dry run feed can be selected from the PLC. When dry run feed is selected via the PLC, the interface signal ”Activate dry run feed” must be set from the PLC user program.		
Signal status 0 or edge evaluation 1 ---> 0	The programmed feedrate is used for traversing. Active after Reset status.		
Application example(s)	Testing a part program with increased feedrate.		
Related to	IS “Dry run feed selected“ (V17000000.6) SD: DRY_RUN_FEED (dry run feed)		

VB32000004 Interface signal	Feed override Signal(s) to channel (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The feed override is set via the PLC in the form of a Gray code.	
Related to	IS "Feed override active" (V32000006.7)	

Switch Position	Code	Feed Override Factor
1	00001	0.0
2	00011	0.01
3	00010	0.02
4	00110	0.04
5	00111	0.06
6	00101	0.08
7	00100	0.10
8	01100	0.20
9	01101	0.30
10	01111	0.40
11	01110	0.50
12	01010	0.60
13	01011	0.70
14	01001	0.75
15	01000	0.80
16	11000	0.85
17	11001	0.90
18	11011	0.95
19	11010	1.00
20	11110	1.05
21	11111	1.10
22	11101	1.15
23	11100	1.20
24	10100	1.20
25	10101	1.20
26	10111	1.20
27	10110	1.20
28	10010	1.20
29	10011	1.20
30	10001	1.20
31	10000	1.20

Table 11-1 Gray coding for feed override

VB32000005 Interface signal	Rapid traverse override Signal(s) to channel (PLC → NCK)	
Edge evaluation: nein	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The rapid traverse override is set via the PLC in the form of a Gray code.	

Switch Position	Code	Rapid Traverse Override Factor
1	00001	0.0
2	00011	0.01
3	00010	0.02
4	00110	0.04
5	00111	0.06
6	00101	0.08
7	00100	0.10
8	01100	0.20
9	01101	0.30
10	01111	0.40
11	01110	0.50
12	01010	0.60
13	01011	0.70
14	01001	0.75
15	01000	0.80
16	11000	0.85
17	11001	0.90
18	11011	0.95
19	11010	1.00
20	11110	1.00
21	11111	1.00
22	11101	1.00
23	11100	1.00
24	10100	1.00
25	10101	1.00
26	10111	1.00
27	10110	1.00
28	10010	1.00
29	10011	1.00
30	10001	1.00
31	10000	1.00

Table 11-2 Gray coding for rapid traverse override

Related to	IS ”Rapid traverse override active“

V32000006.0 Interface signal	Feed blocking Signal(s) to channel (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The signal of one channel is active in all operating modes. • If G33 (thread) is not present, this signal leads to feed blocking of all axes involved in interpolation. All axes are set to standstill without violation of the path contour. Cancellation of feed blocking (0 signal) will continue the interrupted part program. • The position control remains; i.e. the following error is reduced to zero. • I traversing request is provided for an axis for which “Feed Blocking” is provided. the traversing request remains. This traversing request is carried out immediately at the moment when “Feed blocking” is canceled. This is also applicable when the axis is involved in interpolation with other axes.	
Signal status 0 or edge change 1 ---> 0	• The feed is active for all axes of the channel. • If a traversing request (”Traversing command”) is provided for an axis or axis group when “Feed blocking” is canceled, the traversing request is carried out immediately.	
Special cases, errors, ...	Feed blocking with active G33 without effect.	

V32000006.6 Interface signal	Rapid traverse override inactive Signal(s) to channel (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The rapid traverse override from 0 to max. 100 % entered in the PLC interface is active channel–specifically.	
Signal status 0 or. edge change 1 ---> 0	The rapid traverse override entered in the PLC interface is not considered. When rapid traverse override is inactive, the NC internally uses an override factor of 100 %.	
Special cases, errors, ...	Rapid traverse override is disabled when G33 is active.	
Related to	IS ”Rapid traverse override“	

V32000006.7 Interface signal	Feed override active Signal(s) to channel (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The feed override 0 to max. 120 % entered in the PLC interface is active for path feed and thus automatically for the associated axes. In JOG mode, the feed override acts directly on the axes.	
Signal status 0 or edge change 1 ---> 0	The feed override entered in the PLC interface will not be considered. When the feed override is inactive, the NC internally uses an override factor of 100%.	
Special cases, errors, ...	The feed override is inactive when G33 is active.	
Related to	IS ”Feed override“	

V32001000.3 and V32001008.3 Interface signal	Feed Stop (axes in WCS) Signal(s) to channel (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The signal is only active in JOG mode (traversing the axes in WCS). • The signal results in Feed Stop of the respective axis. When an axis is traversing, this signal results in controlled deceleration to standstill (ramp stop). No alarm message is output. • The position control remains; i.e. the following error is reduced to zero. • If for an axis for which “Feed Stop” is present a traversing request is provided, the traversing request remains. This traversing request is carried out immediately at the moment when “Feed Stop” is canceled.	
Signal status 0 or edge change 1 ---> 0	• The feed for the axis is active. • If a traversing request (”traversing command”) is provided for the axis when “Feed Stop” is canceled, the traversing request is carried out immediately.	

11.11.2 Signals to Axis/Spindle

VB380x0000 Interface signal	Feed override (axis–specific) Signal(s) to axis (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The axis–specific feed override is output via the PLC in the form of a Gray code.	

Switch Position	Code	Axial Feed Override Factor
1	00001	0.0
2	00011	0.01
3	00010	0.02
4	00110	0.04
5	00111	0.06
6	00101	0.08
7	00100	0.10
8	01100	0.20
9	01101	0.30
10	01111	0.40
11	01110	0.50
12	01010	0.60
13	01011	0.70
14	01001	0.75
15	01000	0.80
16	11000	0.85
17	11001	0.90
18	11011	0.95
19	11010	1.00
20	11110	1.05
21	11111	1.10
22	11101	1.15
23	11100	1.20
24	10100	1.20
25	10101	1.20
26	10111	1.20
27	10110	1.20
28	10010	1.20
29	10011	1.20
30	10001	1.20
31	10000	1.20

Table 11-3 Gray coding for axis–specific feed override

Related to	IS ”Override active

Note:
The signals for the spindle “Spindle override” (VB38032003) and “Feed override valid for spindle” (V38032001.0) are documented in Section LEERER MERKER.

V380x0001.7 Interface signal	Override active Signal(s) to axis/spindle (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	Feed override active: • The axis–specific feed override from 0 to max. 120 % entered in the PLC interface is taken into account. Spindle override active: • The spindle override from 0 to max. 120 % entered in the PLC is taken into account.	
Signal status 0 or edge change 1 ---> 0	The axis–specific feed override or spindle override is inactive. When the override is inactive, the NC internally uses an override factor of 100 %. Exceptions are the 1st switch position for a Gray–coded interface. The override factors entered in the PLC are used here. For the Gray–coded interface, the value entered in the machine data for the 1st interface is output as the override value.	
Special cases, errors, ...	• In spindle operating mode “Oscillation mode”, the spindle override is always supposed with 100 %. • The spindle override acts on the programmed values before the limits (e.g. G26) become active. • The feed override is inactive when G33 is active.	
Related to	IS ”Feed override“ and IS “Spindle override“	

V380x0004.3 Interface signal	Feed Stop/Spindle Stop (axis–specific) Signal(s) to axis/spindle (PLC → NCK)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ---> 1	The signal is active in all operating modes. Feed Stop: • This signal causes the respective axis to be stopped. When an axis is traversing, this signal results in controlled deceleration to standstill (ramp stop). No alarm message is output. • This signal results in Feed Stop of all path axes involved in interpolation if “Feed Stop” is provided for one of the path axes. In this case, **all axes** are brought to standstill, not violating the path contour. The interrupted part program is continued when “Feed Stop” is canceled. • The position control remains; i.e. the following error is reduced to zero. • If for an axis for which “Feed Stop” is present a traversing request is provided, the traversing request remains. This traversing request is carried out immediately at the moment when “Feed Stop” is canceled. If the axis is involved in interpolation with other axes, this also applies to these axes. Spindle Stop: • The spindle is decelerated to standstill along its acceleration curve. • In positioning mode, the positioning process is interrupted by setting the signal “Spindle Stop”. The above mentioned behavior with refer to single axes applies.	
Signal status 0 or edge change 1 ---> 0	Feed Stop: • The feed for the axis is enabled. • If a traversing request (”traversing command”) is provided for the axis when “Feed Stop” is canceled, the traversing command is carried out immediately. Spindle Stop: • The speed is enabled for the spindle. • If “Spindle Stop” is canceled, the spindle is decelerated to the previous speed setpoint with the acceleration characteristic, or in positioning mode, positioning is continued.	
Application example(s)	Feed Stop: • The traversing movements of the machine axes are not started with “Feed Stop” if, for example, certain operating conditions apply to the axis which do not allow axis movement (e.g. door not closed). Spindle Stop: • in order to carry out tool change	
Special cases, errors, ...		

Tool Compensation

12

Brief description

The SINUMERIK 802S/C control system provide calculation of tool compensation data.

- Length compensation
- Radius compensation
- Saving tool data in tool offset memory
  - Tool coding by T numbers ranging from 0 to 32000
  - Definition of a tool by max. 9 edges
  - The edge is described by tool parameters:
    - Tool type
    - Geometry: Length Wear: Length
    - Geometry: Radius Wear: Radius
    - Edge position (with turning tools)
- Tool change can be selected either immediately with T command or via M6
- Tool radius compensation
  - Compensation active for all interpolation types:
    Linear
    Circular
  - Compensation on external corners can be selected via transition circle (G450) or intersection point of equidistants (G451)
  - Automatic detection of external/internal corners

Note: Detail description - see User Manual “Operation and Programming”.

12.1 Tool

Selecting the tool

A tool is selected in the program by means of the T function. Whether the new tool is changed immediately with the T function, depends on the setting in MD: TOOL_CHANGE_MODE (new tool compensation with M function).

Tool change immediately

MD: TOOL_CHANGE_MODE = 0
The new tool is changed immediately with the T function. This setting is mainly used for turning machines with tool revolver.

Tool change with ”M06”

MD: TOOL_CHANGE_MODE = 1
The new tool is prepared for tool change with the T function. This setting is mainly used for machines with tool magazines in order to bring the tool to tool change position within the main machining time (the machining is not interrupted). M6 is used to remove the old tool from the spindle and load the new tool.

Note

The actual tool change must be performed either by means of the PLC program or manually. The control system calculates the appropriate tool offsets only at the appropriate time.
Note:
If a special tool has been activated, it remains stored as the active tool even beyond the program end and after POWER ON.
If you change a tool manually, you must also enter the change into the control system to make sure that the control system identifies the tool. For example, you can start a block with the new T word in MDA mode.

T value range

For the T function, integer values between T0 (no tool) and T32000 (tool with number 32000) can be used.

Tool compensation

A tool can have up to 9 tool edges. The 9 tool edges are assigned to the D functions D1 to D9.

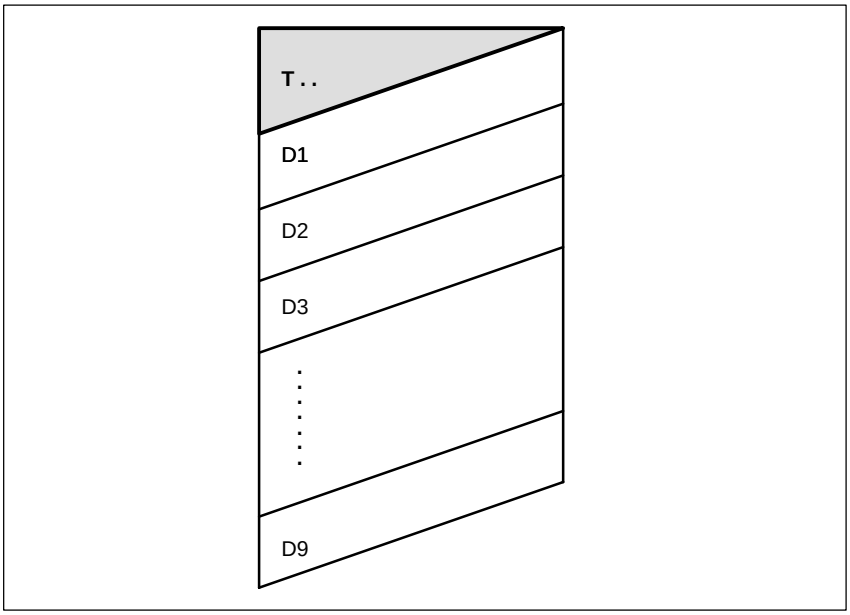


Fig. 12-1 Example of tool T... with 9 cutting edges (D1 to D9)

D function

The tool edge is programmed with D1 (edge 1) to D9 (edge 9). The tool edge always refers to the currently active tool. An active tool edge (D1 to D9) without active tool (T0) is not active. Tool edge D0 deselects all tool compensations of the active tool.

Selecting the edge on tool change

After programming a new tool (new T number) and changing this tool, the following possibilities to select the edge are provided:

1. The edge number is programmed.
2. The edge number is not programmed. D1 is active automatically by default.

Activating the tool compensation

D1 to D9 are used to activate the tool compensation of a tool edge for the active tool. However, tool length compensation and tool radius compensation become active at different moments:

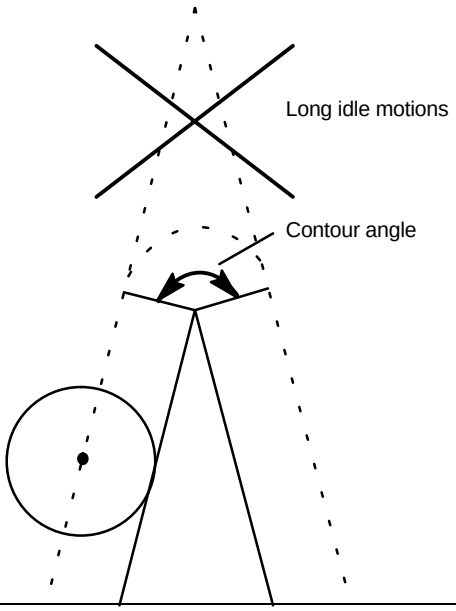
- The tool length compensation (TLC) is realized with the first traversing movement of the axis in which the TLC is to be active. This traversing movement must be a linear interpolation (G0, G1).
- The tool radius compensation (TRC) becomes active by programming G41/G42 in the active plane (G17, G18 or G19). The selection of TRC by G41/G42 may only be performed in a program block with G0 (rapid traverse) or G1 (linear interpolation).

Tool radius compensation

see User's Manual "Operation and Programming", Section "Tool and Tool Compensation"

12.2 Data Description

Machine data

20210 MD number	CUTCOM_CORNER_LIMIT Maximum angle for compensation blocks for tool radius compensation		
Default: 100.0	Min. input limit: 0.0	Max. input limit: 150.0	
Changes effective after Power On	User class: 2/7	Unit: degrees	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	In case of very sharp external corners, long idle motions can occur when G451 is used. This is the reason why an automatic change from G451 (intersection point) to G450 (transition circle) is carried out in the case of very sharp external corners. The contour angle from which this automatic change (intersection point —> transition circle) is carried out can be set by means of this MD.		
			

22550 MD number	TOOL_CHANGE_MODE New tool/tool compensation with M6		
Default: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective after Power On	User class: 2/7	Unit: -	
Data type: BYTE	Valid as from SW version:		
Meaning:	A tool is selected in the program by means of the T function. Whether the new tool is changed immediately with the T function, depends on the setting in the following machine data: **MD: TOOL_CHANGE_MODE = 0**: The new tool is changed immediately with the T function. This setting is mainly used for turning machines with tool revolver. **MD: TOOL_CHANGE_MODE = 1**: The new tool is prepared for change with the T function. This setting is mainly used for machines with tool magazines in order to bring the new tool to the tool change position within the main machining time (the machining is not aborted). M6 is used to remove the old tool from the spindle and to load the new tool into the spindle.		

EMERGENCY STOP

13

Brief description

Standard EN 292–2

In accordance with the basic safety requirements set forth by the EU Machine Guidelines on EMERGENCY STOP, which are contained in Section 6.1.1 (EN 292–2), machines must be equipped with an EMERGENCY STOP system.

Countries where the above mentioned guideline is not applicable must adhere to the guidelines for safety requirements on EMERGENCY STOP of the relevant country.

Exceptions

No EMERGENCY STOP is required for

- machines on which an EMERGENCY STOP system would not reduce the risk or hazard because the EMERGENCY STOP system would neither reduce the stop time, nor the required actions would reduce the risk.
- machines which can be carried by hand, or for hand–controlled machines.

EMERGENCY STOP in the control system

The control system assists the machine manufacturer in the realization of the EMERGENCY STOP function by the following:

- The EMERGENCY STOP button is easily accessible arranged on the machine control panel and marked by a red rim.
- Red EMERGENCY STOP button with positive opening operation and provided with a mechanically operated self–locking mechanism.
- Triggering of EMERGENCY STOP sequence in the NC via the PLC input.
- The EMERGENCY STOP sequence in the NC decelerates all axes and spindles as fast as possible.
- In case of EMERGENCY STOP, all machine functions controlled by the PLC can be in a settable, safe condition.
- EMERGENCY STOP status cancellation by unlocking the EMERGENCY STOP button is not possible.
  Resetting the control device does not result in restart.

13.1 General

Important

The machine manufacturer is herewith referred to the international and national standards (see notes on the standards below in the text). The SINUMERIK 802S supports the machine manufacturer in the realization of the EMERGENCY STOP function according to the conventions made in this Functional Description. The realization of the EMERGENCY STOP function (triggering, sequence, acknowledgement) is the sole responsibility of the machine manufacturer.

Note

For the EMERGENCY STOP function, you are referred, in particular, to the following standards:

- EN 292 Part 1
- EN 292 Part 2
- EN 418
- EN 60204 Part 1:1992 Section 10.7

EMERGENCY STOP function

EN 418: EMERGENCY STOP is a function

- intended to avert or reduce impending or existing danger for persons, as well as damage to the machine or material.
- triggered as a single action by one person if normal stop is not the appropriate action for the intended purpose.

In the sense of EN 418, hazards can be caused by:

- functional inconsistencies (malfunctions of the machine, inacceptable properties of the material, operator faults, ...).
- normal operation.

13.2 EMERGENCY STOP Tripping Mechanism

Standards EN 418

Acc. to EN418, EMERGENCY STOP tripping mechanisms must be arranged within the reach of the operator or any other persons who consider the operation of the EMERGENCY STOP tripping mechanisms necessary. Among other types, the following tripping mechanisms can be used:

- Mushroom buttons (push–button–operated switches)
- Wires/wire ropes, lines, bars
- Knobs
- in special cases: foot–operated switches without protection cover

All EMERGENCY STOP tripping mechanisms must be mechanically self–locking and be arranged within reach.

EMERGENCY STOP button

The Siemens machine control panel (MCP) for 802S is provided with a mushroom button (push–button–operated switch with positive opening operation), further called EMERGENCY STOP button.
Reference: Technical Manual, Start–Up Instructions

EMERGENCY STOP on NC

The actuation of the EMERGENCY STOP button or a signal directly derived from it must be passed on to the control (PLC) as a PLC input. In the PLC user program, this PLC input must be passed on to the NC to the IS ”EMERGENCY STOP” (V26000000.1).
Resetting of the EMERGENCY STOP button or the signal directly derived from it must be passed on to the control (PLC) as a PLC input. In the PLC user program, this PLC input must be passed on to the NC to the IS ”Acknowledge EMERGENCY STOP“ (V26000000.2).

13.3 EMERGENCY STOP Sequence

Standard EN 418

After actuating the EMERGENCY STOP tripping mechanism, the EMERGENCY STOP system must operate such that any impending hazards are averted or reduced as best as possible.
“As best as possible“ means that the the optimum deceleration rate and the appropriate stop category (defined in EN 60204) based on a risk evaluation are selected.

Sequence in the NC

The defined sequence (acc. to EN 418) of the internal functions to bring the system to the EMERGENCY STOP status in the control system is as follows:

1. Part program execution is interrupted; all axes and spindles are decelerated. The spindle and the axes is decelerated along a deceleration ramp defined by MD AX_EMERGENCY_STOP_TIME; the stepper motor axes are decelerated along a fixed, internal deceleration ramp.
2. IS ”READY” (V31000000.3) is reset.
3. IS ”EMERGENCY STOP active” (V27000000.1) is set.
4. Alarm 3000 is set.
5. After the spindle–specific time which can be set in MD 36620: SERVO_DISABLE_DELAY_TIME (servo enable shutdown delay) has elapsed, servo enable is switched off. It must be made sure that SERVO_DISABLE_DELAY_TIME is at least as high as set in AX_EMERGENCY_STOP_TIME.

Sequence on the machine

The EMERGENCY STOP sequence on the machine is exclusively defined by the machine manufacturer. In conjunction with the sequence in the NC, the following should be taken into account:

- The sequence in the NC is started with the IS ”EMERGENCY STOP” (V26000000.1). After the axes and spindles have come to standstill, the power supply must be disconnected acc. to EN418.

Important

The disconnection of the power supply is the sole responsibility of the machine manufacturer.

- The sequence in the NC has no influence on the PLC interface modules (digital outputs). If individual outputs are to have a certain status in case of EMERGENCY STOP, the machine manufacturer must provide for the respective functions in the PLC user program.

Important

If you wish the sequence in the NC to be performed for EMERGENCY STOP not as defined, the IS “EMERGENCY STOP” (V26000000.1) may not be set prior to the EMERGENCY STOP status defined by the machine manufacturer in the PLC user program. As long as the IS ”EMERGENCY STOP” is not yet set and no other alarm is present, all IS are active in the NC. Any manufacturer–specific EMERGENCY STOP status is thus possible.

13.4 EMERGENCY STOP Acknowledgement

Standard EN 418

The resetting of the EMERGENCY STOP tripping mechanism may only be possible as a result of an action on the EMERGENCY STOP tripping mechanism. The resetting of the EMERGENCY STOP tripping mechanism alone may not trigger a restart command. Restart of the machine may not be possible until all actuated EMERGENCY STOP tripping mechanisms have been reset manually, separately and deliberately.

Acknowledge EMERGENCY STOP

The EMERGENCY STOP status is only reset if first the IS ”Acknowledge EMERGENCY STOP“ (V26000000.2) and then the IS ”Reset” (V30000000.7) are set. When doing this, make sure that the IS ”Acknowledge EMERGENCY STOP“ and the IS ”Reset” must be set together and at least as long as the IS ”EMERGENCY STOP active” (V27000000.1) is reset (see Fig. 13-1).

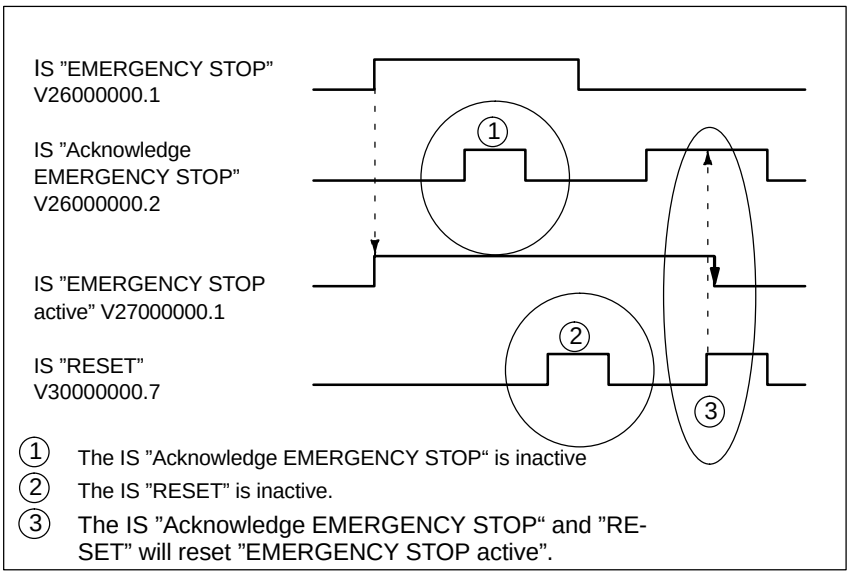


Fig. 13-1 Resetting EMERGENCY STOP

Resetting the EMERGENCY STOP status:

- resets the IS ”EMERGENCY STOP active”;
- activates servo enable;
- sets the IS ”Position control active“;
- sets the IS ”READY”
- deletes alarm 3000;
- aborts part program execution.

PLC interface system

The PLC interface system must be set by the PLC user program to such a condition that the operation of the machine is possible.

Reset

The IS ”Reset” (30000000.7) alone will not reset the EMERGENCY STOP status (see illustration above).

Power On

Power On will delete the EMERGENCY STOP status except the IS ”EMERGENCY STOP” (V26000000.1) is still active.

13.5 Data Description

36620 MD number	SERVO_DISABLE_DELAY_TIME Servo disable delay time		
Default: 0.1	Min. input limit: 0.02	Max. input limit: 1000	
Change valid after NEW_CONF	Protection level: 2/7	Unit: s	
Data type: DOUBLE	Valid as from SW version:		
Meaning:	Maximum delay for Servo Disable after faults. The speed enable (servo enable) of the drive is canceled internally in the control system after the set delay time at the latest provided the axis / spindle is moving. The entered delay time is active due to the following events: • in case of errors resulting in an immediate stop of the axes • if the IS “Servo enable” is canceled from the PLC. Once the actual speed reaches the standstill range (MD: STANDSTILL_VELO_ TOL), Servo Enable is canceled for the drive. The time should be set such that the axis/spindle can come to a standstill from its maximum traversing velocity/speed. If the axis/spidle is at a standstill, Servo Enable for the drive is canceled immediately.		
Application example(s)	The speed control of the drive should be maintained for this time to ensure that the axis/spindle can come to a standstill from its maximum traversing velocity/speed. For this time, Servo Disable should be delayed for an axis/spindle moving.		
Special cases, errors, ...	**CAUTION:** If the servo disable delay is set to low, servo enable is already canceled although the axis is still traversing. In this case, it is suddenly stopped with setpoint 0. For this reason, the time defined in this MD should be greater than the time of the brake ramp under error conditions (MD: AX_EMERGENCY_STOP_TIME).		
Related to	IS ”Servoce enable” (V380x0002.1) MD: AX_EMERGENCY_STOP_TIME (time of brake ramp under error conditions)		

13.6 Signal Description

V26000000.1 Interface signal	NOT AUS Signal(s) to NC (PLC ——> NC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 —> 1	The NC is set to the EMERGENCY STOP condition, starting the EMERGENCY STOP sequence in the NC.	
Signal status 0 bzw. edge change 1 —> 0	● The NC is not in the EMERGENCY STOP condition. ● The EMERGENCY STOP condition is (still) active, but can be reset with IS: ”Acknowledge EMERGENCY STOP” and IS ”Reset”.	
Related to	IS ”Acknowledge EMERGENCY STOP” (V26000000.2) IS ”EMERGENCY STOP active” (V27000000.1)	

V26000000.2 Interface signal	Acknowledge EMERGENCY STOP Signal(s) to NC (PLC ——> NC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 —> 1	The EMERGENCY STOP condition will only be reset if first the IS “Acknowledge EMERGENCY STOP” is set and then the IS ”Reset” (V30000000.7). In this context, it should be taken into account that the IS “Acknowledge EMERGENCY STOP” and the IS “Reset” must be set together at least until the IS EMERGENCY STOP active” (V26000000.1) has been reset. Resetting the EMERGENCY STOP condition has the following effects: ● The IS ”EMERGENCY STOP active” is reset. ● Servo Enable is provided. ● The IS ”Position control active” is set. ● The IS ”READY” is set. ● The alarm 3000 is deleted. ● The part program execution is canceled.	
Related to	IS ”EMERGENCY STOP” (V26000000.1) IS ”EMERGENCY STOP active” (V27000000.1) IS ”Reset” (V30000000.7)	

V27000000.1 Interface signal	NOT AUS aktiv Signal(s) to NC (PLC ——> NC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 —> 1	The NC is in the EMERGENCY STOP condition.	
Related to	IS ”EMERGENCY STOP” (V26000000.1) IS ”Acknowledge EMERGENCY STOP” (V26000000.2)	

Various Interface Signals

14

Brief description

This Description of Functions describes the functionality of various interface signals, which are of general significance and have not yet been described in any other descriptions of functions before.

14.1 General

Interfaces

Signals and data between the PLC user program and

- NCK (numerical control kernel)
- MMC (display unit)
- MCP (machine control panel)

are exchanged via different data areas. The PLC user program need not care for the exchange. From user's point of view, this is carried out automatically.

Cyclic signal exchange to NCK

The control and status signals of the PLC/NCK interface are updated cyclically.

These can be divided into the following groups (see Fig. 14-1):

- General signals
- Operating mode signals
- Channel signals
- Axis/spindle signals

The interface structure is described in detail in:
References: Start–Up Guide, Chapter "PLC Start–Up"

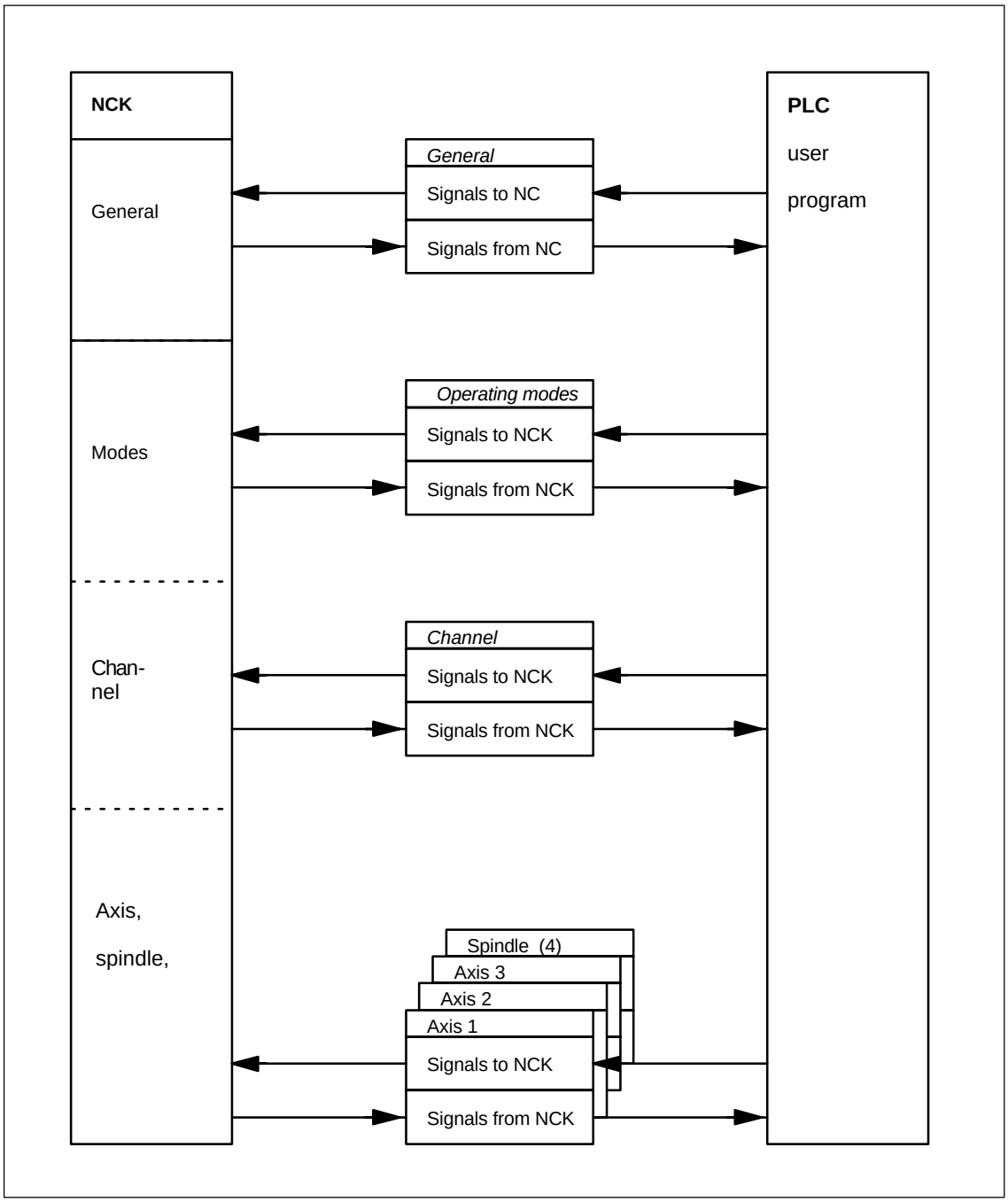


Fig. 14-1 PLC/NCK interface

14.2 Signals from PLC to NCK

Access rights

The access to programs, data and functions is user–oriented protected via 8 hierarchical user classes (protection levels). These are divided into:

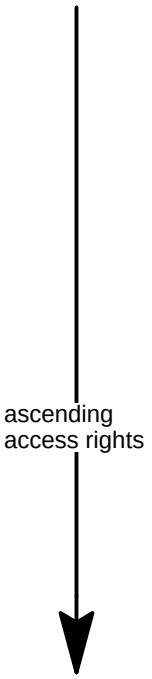
- 4 password levels for Siemens, machine manufacturer and end user
- 4 password levels for end user (interface signals V26000000.4 to .7)

This provides a multi–stage safety concept to regulate the access rights.

see also Start–Up Instructions, Section “Access Levels”

Table 14-1 Write protection

User Class	Method	User	Access to Examples
0	Pass–word	SIEMENS	All functions, programs and data
1	Pass–word	SIEMENS/ Machine manuf.	Defined functions, programs and data; e.g.: Enter options
2	Pass–word	Machine manuf.	Defined functions, programs and data; e.g.: major part of machine data
3	Pass–word	End user: Service	Assigned functions, programs and data
4	**IS V2600 0000.7**	End user: Programmer Setter	Lower than user class 0 to 3; defined by machine manufacturer or end user
5	**IS V2600 0000.6**	End user: qualified operator,	lower than user class 0 to 3; defined by end user not programmed
6	**IS V2600 0000.5**	End user: qualified operator who does not program	Example: only program selection, tool wear input and input of zero offsets
7	**IS V2600 0000.4**	End user: trained operator	Example: no inputs and no program selection possible, only machine control panel can be operated



Request axis actual values V26000001.1

(software version 3.1 and higher)
The cyclic provision of the current actual value positions is requested for all axes in the range VD570x0000.

Request distances to go for the axes V26000001.2

(software version 3.1 and higher)
The cyclic provision of the distances currently to go is requested for all axes in the range VD570x0004.

Deletion of the distance to go (channel–specifically) **V32000006.2**

(software version 3.1 and higher)
The IS “Delete distance to go” (channel–specifically) is only effective in the AUTOMATIC mode; it is effective there for all axes programmed in the block.
With the rising edge of the interface signal, these axes are stopped with ramp stop; the following error (if any) will be reduced to zero. The distance to go remaining up to the end of the block (distance to go along the path) is deleted; then, the next program block is started for execution.

Note:
After the axes have been stopped using the IS “Delete distance to go”, the block preparation for the subsequent program block is carried out using the new positions. Thus, after “Delete distance to go”, the axes will travel along another contour than originally defined in the part program.
It can be achieved by programming G90 in the block following after “Delete distance to go” that at least the programmed absolute position is approached. In contrast, the position which has originally been defined with G91 in the part program will no longer be reached in the subsequent block.

V380x0001.3 Interface signal	Axis/spindle lock Signal(s) to axis/spindle (PLC → NC)	
Edge evaluation: no	Signal(s) updated: cyclically	Signal(s) valid as from SW version:
Signal status 1 or edge change 0 ——> 1 – Axis lock	The IS “Axis/spindle lock” can only be used for testing purposes. (test status) If the IS “Axis lock” is provided, no position setpoints are output to the position controller of this axis; the traversing movement of this axis is thus blocked. The position control loop remains closed and the remaining following error is compensated. If an axis is traversed with axis lock, the actual position display will show the set position and the velocity actual–value position even without real movement of the machine axis. RESET (IS V30000000.7) will set the position actual value display to the real actual value of the machine. Traversing commands for this axis are still output to the PLC. If the interface signal is canceled, the associated axis can be traversed normally again. If the interface signal “Axis lock” is provided for a traversing axis, the axis is stopped with deceleration stop.	
– Spindle lock	If the IS “Spindle lock” is provided, analogously to axis lock in control mode, no speed setpoints, or in control mode, no position actual values are output to the position controller any more. The motion of the spindle is thus blocked. The speed actual value display displays the speed setpoint. Spindle lock can only be canceled by ”Reset” or M2 and restart of the program. If the interface signal “Spindle lock” is provided when the spindle is rotating, the spindle is stopped as defined by its acceleration characteristic.	
Signal status 0 or edge change 1 ——> 0	(normal position). The position and speed setpoints are transferred to the position controller cyclically. Cancellation of “Axis/spindle lock” (edge change 1 → 0) is only active if the spindle is on standstill (i.e. no interpolation setpoint is present any longer). The new motion will start with the new setpoints (e.g.: new program block with motion settings in AUTOMATIC mode). Note: different actual values between simulated and real axis!	
Application example(s)	The interface signals ”Axis lock” and ”Spindle lock“ are used for testing NC part programs. The machine axes and spindles may not carry out any traversing or rotational movement.	
Special cases, errors, ...	If “Axis/spindle lock is provided for an axis/spindle, the interface signals “Servo enable”, “Feed/ spindle stop” and “Hardware limit switches” (if any) are not active for the brakes of axis/spindle.	
Related to	IS ”Program test active” (V3300000001.7)	

Follow–up mode V380x0001.4

software version 3.1 and higher)

If an axis/spindle is in the follow–up mode, its setpoint position will always follow the current actual value position. With follow–up mode, the position setpoint value is not specified by the interpolator, but derived from the current actual position. Since the position actual value of the axis is continued to be acquired, it is not necessary to re–reference the axis after cancellation of the follow–up mode.
Zero–speed, clamping and positioning monitoring are not effective in the follow–up mode.

Activation:
The IS “Follow–up mode” is only relevant if servo enable is cancelled for the drive (e.g. via the IS “Servo enable” = 0 signal or internally in the control system due to a fault) or if servo enable is provided once more.

IS ”Follow–up mode“ = 1:
If the IS ”Servo enable“ is cancelled, the position setpoint value of the appropriate axis will follow the actual value continuously. This status is dislayed via the IS “Follow–up active” (V390x0001.3) transmitted to the PLC. If then the IS “Servo enable” is set again, repositioning (REPOSA: approach with all axes along a straight line) is carried out (provided a part program is active) internally in the control system to the position last programmed.
Otherwise (no part program active), the axis motion will start at the possibly changed, new actual position.

IS ”Follow–up mode” = 0:
If the IS “Servo enable” is canecelled, the old position setpoint is kept. If the axis is pressed out of its position, a following error will result between position setpoint and position actual values which will be compensated when setting the IS ”Servo enable“. The axis motion will start at the setpoint position that existed before “Servo enable” was cancelled.
The IS ”Follow–up active” (V390x0001.3) will not be set here.
Clamping or zero–speed monitoring is active.

Position measuring system 1 V380x0001.5

The spindle can be equipped with a position measuring system. In this case, the signal for the spindle must be set.
This signal is always required for axes with analog drives or stepper motor axes even if no measuring systems are connected for stepper motor axes.

Servo enable V380x0002.1

When servo enable for the drive is provided, the position–control loop for the axis/spindle is closed. The axis/spindle is thus in position/control mode.
When servo enable is canceled, the position–control loop is opened, and, with delay, also the speed–control loop of the axis/spindle.
The IS ”Position controller active“ (V390x0001.5) is set to 0 signal (feed back).

Activation:
Setting and resetting of servo enable for the drive can be provided from the following places:

1. from the PLC user program by means of the interface signal ”Servo enable“ (normal case)

Application: servo enable canceled before clamping the axis/spindle.

2. In case of various faults on the machine, the drive, the position measuring system or the control system, servo enable is disabled in the control system internally.

Application: Due to faults, the moving axes must be stopped by rapid stop.

3. internally in the control system in case of the following events:

- EMERGENCY STOP is present on the PLC interface.

Servo Disable for a moving axis/spindle:

- The spindle/analog axis is decelerated to a standstill by quick stop with consideration of the MD: AX_EMERGENCY_STOP_TIME (duration of deceleration ramp in case of fault conditions). Then alarm 21612 ”Servo enable reset during the motion” is output.
- The position control loop of axis/spindle is opened. Feedback to the PLC with the IS ”Position controller active“ (V390x0001.5) = 0 signal. In addition, the servo enable delay timer (MD: SERVO_DISABLE_DELAY_TIME (servo enable shutdown delay)) is started.
- Once the actual velocity has reached the standstill range, servo enable is canceled. Feedback to the PLC with the IS ”Speed controller active” (V390x0001.6) = 0 signal. Servo enable for the drive is disabled at least after the time set in MD: SERVO_DISABLE_DELAY_TIME has elapsed.

CAUTION: If servo enable shutdown delay is set too low, servo enable is canceled even when the axis/spindle is still traversing. It is then suddenly stopped with setpoint zero.

- The position actual value of the axis/spindle is still acquired by the control system.

This status of the axis/spindle can only be changed after ”Reset”.

Interpolatory axis grouping:
All axes involved in interpolation are stopped once servo enable is canceled for one of the axes involved in interpolation.

The axes are stopped as described above. All axes of the geometrical axis grouping are stopped with rapid stop. Furthermore, alarm 21612 ”Servo enable reset during the motion” is signaled. In this case, the NC program can no longer be continued.

Deletion of the distance to go /spindle reset (axis–specifically) **V380x0002.2**

(software version 3.1 and higher, extended to “Delete distance to go” – axis–specifically)
The effect of the IS on the spindle (”Spindle reset”) is described in Section 9.7.
Effect with axes: Delete distance to go – axis–specifically
The effect is mode–dependent.
With JOG:
If the interface signal is provided for one axis (edge transition 0 —>1),
then this axis is stopped by ramp stop and its deletion to go deleted. The following error (if any) will be reduced to zero.
With AUTOMATIC and MDA:
The rising edge of the interface signal (IS) is only effective with axes which are not integrated in the geometrical grouping (interpolatory axis grouping).
SINUMERIK 802S/C, however, possesses axes which are integrated in the geometrical grouping.
Therefore, the IS “Delete distance to go” – axis–specifically will be ignored.
In this case, use the IS “Delete distance to go” –channel–specifically (V32000006.2).

Rotation monitoring (stepper motor) V380x5000.0

For rotation monitoring, the stepper motor requires a BERO (proximity switch). Parallel connection with the BERO for referencing (see Section “Reference–point approach“) or using the BERO for rotation monitoring is possible.
For this reason, rotation monitoring is not active during reference–point approach. The BERO signals are used for reference–point approach.
However, when rotation monitoring is active no signals may be provided from the reference BERO.
The monitoring function checks whether the axis has reached the preset path increments with the set tolerance during one motor revolution.
Important axis machine data for rotation monitoring:
MD:BERO_CYCLE and MD:BERO_EDGE_TOL

In case of error, the IS “Rotation monitoring error“ (V390x5000.0) is set.

see also Start–Up Instructions, Section “Axis Start–Up”

14.3 Signals from NCK to PLC

Drive ready V27000002.6

The PLC receives a signal that all connected drives are ready for operation via the NCK.

NCK alarm present V27000003.0

The control system signals to the PLC that at least one NCK alarm is present. The channel–specific interface (V33000004.7) can be interrogated whether this has caused a stop in processing.

Air temperature alarm V27000003.6

Ambient temperature or fan monitoring has responded.

Channel–specific NCK alarm present V33000004.6

The control system signals to the PLC than at least one NCK alarm is present for the channel. In which way the current program execution has been interrupted or aborted as a result of this, can be derived from the IS “NCK alarm with processing stop present” (V33000004.7).

NCK alarm with processing stop present V33000004.7

The control system signals to the PLC that at least one NCK alarm is present for the channel, which has interrupted or aborted the execution of the current program (processing stop).

Follow–up active V390x0001.3

FI(software version 3.1 and higher)
Follow–up mode is active for this axis
(”Follow–up” in details: see at IS “Follow–up mode” (V380x0001.4))

Axis/spindle on standstill V390x0001.4

The current actual velocity of the axis or the actual speed of the spindle is within the range defined as the standstill. This range is defined by MD: STANDSTILL_VELO_TOL (max. velocity/speed for signal ”Axis/spindle on standstill“).

Position controller active V390x0001.5

The position controller for the axis/spindle is closed; the position control is active.

Speed controller active V390x0001.6

The speed controller for the axis/spindle is closed; speed control is active.

Current controller active V390x0001.7

The current controller for the axis/spindle is closed; current control is active.

Lubrication pulse V390x1002.0

The IS ”Lubrication pulse“ is sent from the NCK and changes its status once the axis/spindle has covered a distance longer than entered in MD: LUBRICATION_DIST (distance to be traversed for lubrication from PLC).

Rotation monitoring error V390x5000.0

The signal “Rotation monitoring error” is set if the stepper motor was incorrectly controlled even if “Rotation monitoring” (V380x5000.0) is not active.
The user must stop the drive safely.
In this case, the reference point gets lost. To continue work, re–referencing is required.

Axis actual values VD570x0000

(software version 3.1 and higher)
If cyclic processing of the current actual value position has been requested for all axes via the IS “Request axis actual values” (V26000001.1), then the actual position is delivered for the axis concerned (data format: 4–byte floating point = FLOAT).

Distances to go by the axes VD570x0004

(software version 3.1)
If cyclic processing of the current distances to go has been requested for all axes via the IS “Request distances to go by the axes” (V26000001.2), then the distance to go is delivered for the axis concerned (data format: 4–byte floating point = FLOAT).

14.4 Signals from PLC to MMC

Key lock V19005000.2

The IS ”Key lock“ can be used to lock (”1” signal) or unlock (”0” signal).

List of Interface Signals

15

Brief description

The following contains an overview of the interface signals exchanged between
NCK/PLC,
MMC/PLC and
the machine control panel (MCP)/PLC.
For more detail information, the list contains references on further documentation/sections:

1/xx :Section xx of this Functional Description
2/ :Section “Start–Up of PLC“ of Start–Up Instructions

15.1 Interface Signals

Interface signal	Name	Ref./Section
General (PLC –> NCK)		
V26000000.1	EMERGENCY STOP	1/13
V26000000.2	Acknowledge EMERGENCY STOP	1/13
V26000000.7 to .4	User class 4 to 7	1/14
V26000001.1	Request axis actual values	1/14
V26000001.2	Request distances to go by the axes	1/14
General (NCK –> PLC)		
V27000000.1	EMERGENCY STOP active	1/13
V27000002.6	Drive ready	1/14
V27000003.0	NCK alarm present	1/14
V27000003.6	Air temperature alarm present	1/14
Operating modes (PLC –> NCK)		
V30000000.0	AUTOM.	1/5
V30000000.1	MDA	1/5
V30000000.2	JOG	1/5
V30000000.4	Mode change blocked	1/5
V30000000.7	Reset	1/5,13
V30000001.0	Machine function: TEACH IN	1/5, 8
V30000001.2	Machine function: REF	1/5, 8
Operating modes (NCK –> PLC)		
V31000000.0	AUTOM. active	1/5
V31000000.1	MDA active	1/5
V31000000.2	JOG active	1/5
V31000000.3	READY	1/5
V31000001.0	Machine function: TEACH IN active	1/5, 8
V31000001.2	Machine function: REF active	1/5, 8
Channel (PLC –> NCK)		
V32000000.4	Activate Single Block	1/5
V32000000.5	Activate M01	1/5
V32000000.6	Activate dry run feed	1/5
V32000001.0	Activate referencing	1/8
V32000001.7	Activate program test	1/5
V32000002.0	Activate Block Skip	1/5
VB32000004	Feed override (override value)	1/11
VB32000005	Rapid traverse override (override value)	1/11
V32000006.0	Feed lock	1/11
V32000006.1	Read–in disable	1/5

Channel (PLC –> NCK)		
V32000006.2	Delete distance to go – channel–specifically	1/14
V32000006.4	Program level abortion	1/5
V32000006.6	Feed override enabled	1/11
V32000006.7	Feed override enabled	1/11
V32000007.0	NC Start inhibited	1/5
V32000007.1	NC Start	1/5
V32000007.2	NC Stop at block end	1/5
V32000007.3	NC Stop	1/5
V32000007.4	NC Stop Axes plus spindle	1/5
V32001000.0	Axis 1 in WCS: Activate handwheel 1	1/4
V32001000.1	Axis 1 in WCS: Activate handwheel 2	1/4
V32001000.3	Axis 1 in WCS: Feed Stop	1/11
V32001000.4	Axis 1 in WCS: Traversing key lock	1/4
V32001000.5	Axis 1 in WCS: Rapid traverse override	1/4
V32001000.6	Axis 1 in WCS: Traversing key –	1/4
V32001000.7	Axis 1 in WCS: Traversing key +	1/4
V32001001.0	Axis 1 in WCS: Machine function 1 INC	1/4
V32001001.1	Axis 1 in WCS: Machine function 10 INC	1/4
V32001001.2	Axis 1 in WCS: Machine function 100 INC	1/4
V32001001.3	Axis 1 in WCS: Machine function 1000 INC	1/4
V32001001.6	Axis 1 in WCS: continuous	1/4
V32001004.0	Axis 2 in WCS: Activate handwheel 1	1/4
V32001004.1	Axis 2 in WCS: Activate handwheel 2	1/4
V32001004.3	Axis 2 in WCS: Feed Stop	1/11
V32001004.4	Axis 2 in WCS: Traversing key lock	1/4
V32001004.5	Axis 2 in WCS: Rapid traverse override	1/4
V32001004.6	Axis 2 in WCS: Traversing key –	1/4
V32001004.7	Axis 2 in WCS: Traversing key +	1/4
V32001005.0	Axis 2 in WCS: Machine function 1 INC	1/4
V32001005.1	Axis 2 in WCS: Machine function 10 INC	1/4
V32001005.2	Axis 2 in WCS: Machine function 100 INC	1/4
V32001005.3	Axis 2 in WCS: Machine function 1000 INC	1/4
V32001005.6	Axis 2 in WCS: continuous	1/4
V32001008.0	Axis 3 in WCS: Activate handwheel 1	1/4
V32001008.1	Axis 3 in WCS: Activate handwheel 2	1/4
V32001008.3	Axis 3 in WCS: Feed Stop	1/11
V32001008.4	Axis 3 in WCS: Traversing key lock	1/4
V32001008.5	Axis 3 in WCS: Rapid traverse override	1/4
V32001008.6	Axis 3 in WCS: Traversing key –	1/4
V32001008.7	Axis 3 in WCS: Traversing key +	1/4

Channel (PLC –> NCK)		
V32001009.0	Axis 3 in WCS: Machine function 1 INC	1/4
V32001009.1	Axis 3 in WCS: Machine function 10 INC	1/4
V32001009.2	Axis 3 in WCS: Machine function 100 INC	1/4
V32001009.3	Axis 3 in WCS: Machine function 1000 INC	1/4
V32001009.6	Axis 3 in WCS: continuous	1/4
Channel (NCK –> PLC)		
V33000000.5	M0/M1 active	1/5
V33000001.0	Referencing active	1/8
V33000001.2	Revolutional feed active	1/11
V33000001.4	Block search active	1/5
V33000001.5	M2/M30 active	1/5
V33000001.7	Program test active	1/5
V33000003.0	Program status: Running	1/5
V33000003.1	Program status: Waiting	1/5
V33000003.2	Program status: Stopped	1/5
V33000003.3	Program status: Interrupted	1/5
V33000003.4	Program status: Aborted	1/5
V33000003.5	Channel status: Active	1/5
V33000003.6	Channel status: Interrupted	1/5
V33000003.7	Channel status: Reset	1/5
V33000004.2	All axes referenced	1/8
V33000004.3	All axes on standstill	1/2
V33000004.6	Channel–specific NCK alarm present	1/14
V33000004.7	NCK alarm with processing stop present	1/14
V33001000.0	Axis 1 in WCS: Handwheel 1 active	1/4
V33001000.1	Axis 1 in WCS: Handwheel 2 active	1/4
V33001000.6	Axis 1 in WCS: Traversing command minus	1/4
V33001000.7	Axis 1 in WCS: Traversing command plus	1/4
V33001001.0	Axis 1 in WCS: Machine function 1 INC	1/4
V33001001.1	Axis 1 in WCS: Machine function 10 INC	1/4
V33001001.2	Axis 1 in WCS: Machine function 100 INC	1/4
V33001001.3	Axis 1 in WCS: Machine function 1000 INC	1/4
V33001001.6	Axis 1 in WCS: continuous	1/4
V33001004.0	Axis 2 in WCS: Handwheel 1 active	1/4
V33001004.1	Axis 2 in WCS: Handwheel 2 active	1/4
V33001004.6	Axis 2 in WCS: Traversing command minus	1/4
V33001004.7	Axis 2 in WCS: Traversing command plus	1/4
V33001005.0	Axis 2 in WCS: Machine function 1 INC	1/4
V33001005.1	Axis 2 in WCS: Machine function 10 INC	1/4
V33001005.2	Axis 2 in WCS: Machine function 100 INC	1/4

Channel (NCK –> PLC)		
V33001005.3	Axis 2 in WCS: Machine function 1000 INC	1/4
V33001005.6	Axis 2 in WCS: continuous	1/4
V33001008.0	Axis 3 in WCS: Handwheel 1 active	1/4
V33001008.1	Axis 3 in WCS: Handwheel 2 active	1/4
V33001008.6	Axis 3 in WCS: Traversing command minus	1/4
V33001008.7	Axis 3 in WCS: Traversing command plus	1/4
V33001009.0	Axis 3 in WCS: Machine function 1 INC	1/4
V33001009.1	Axis 3 in WCS: Machine function 10 INC	1/4
V33001009.2	Axis 3 in WCS: Machine function 100 INC	1/4
V33001009.3	Axis 3 in WCS: Machine function 1000 INC	1/4
V33001009.6	Axis 3 in WCS: continuous	1/4
V25000000.0	Decoded M function 0–99 modified	1/10
V25000001.4	T function 1 modified	1/10
V25001000.0 to V25001012.3	Dynamic M functions: M0 to M99	1/10
VB25002000 to VB25002003	T function 1 (4–byte value)	1/10
Axis/spindle (PLC –> NCK)		
VB380x000	Feed override (override value)	1/11
V380x0001.3	Axes/spindle lock	1/14
V380x0001.4	Follow–up mode	1/14
V380x0001.5	Position measuring system 1	1/14
V380x0001.7	Override enable	1/11
V380x0002.1	Servo enable	1/14
V380x0002.2	Delete distance to go / spindle reset	1/9, 14
V380x0002.2	Clear distance to go/Spindle Reset	1/9
V380x0002.3	Clamping process running	1/1
V380x0003.6	Velocity/spindle speed limitation	1/1
V380x0004.0	Activate handwheel 1	1/4
V380x0004.1	Activate handwheel 2	1/4
V380x0004.3	Feed Stop/Spindle Stop	1/11
V380x0004.4	Traversing key lock	1/4
V380x0004.5	Rapid traverse override	1/4
V380x0004.6	Traversing key minus	1/4
V380x0004.7	Traversing key plus	1/4
V380x0005.0	Machine function 1 INC	1/4
V380x0005.1	Machine function 10 INC	1/4
V380x0005.2	Machine function 100 INC	1/4
V380x0005.3	Machine function 1000 INC	1/4
V380x0005.6	Machine function continuous	1/4
V380x1000.0	Hardware limit switch minus	1/1
V380x1000.1	Hardware limit switch plus	1/1

Axis/spindle (PLC –> NCK)		
V380x1000.2	2nd software limit switch minus	1/1
V380x1000.3	2nd software limit switch plus	1/1
V380x1000.7	Reference–point approach delay	1/8
V38032000.0 to .2	Spindle: Actual gear stage A to C	1/9
V38032000.3	Spindle: Gear stage changed	1/9
V38032001.0	Spindle: Feed override valid for spindle	1/9
V38032001.6	Spindle: Invert M3/M4	1/9
V38032002.4	Spindle: Oscillation by PLC	1/9
V38032002.5	Spindle: Oscillation speed	1/9
V38032002.6	Spindle: Set direction of rotation CW	1/9
V38032002.7	Spindle: Set direction of rotation CCW	1/9
VB38032003	Spindle: Spindle override (override value)	1/9
V380x5000.0	Stepper motor: Speed monitoring	1/14
Axis/spindle (NCK –> PLC)		
V390x0000.0	Spindle – no axis	1/9
V390x0000.2	Encoder frequency exceeded 1	1/9
V390x0000.4	Referenced/synchronized 1	1/8
V390x0000.6	Position reached with exact stop	1/2
V390x0000.7	Position reached with exact stop fine	1/2
V390x0001.3	Follow–up active	1/14
V390x0001.4	Axis/spindle on stop ($n < n_{min}$)	1/14, 9
V390x0001.5	Position controller active	1/14
V390x0001.6	Speed controller active	1/14
V390x0001.7	Current controller active	1/14
V390x0004.0	Handwheel 1 active	1/4
V390x0004.1	Handwheel 2 active	1/4
V390x0004.6	Traversing command minus	1/4
V390x0004.7	Traversing command plus	1/4
V390x0005.0	Active machine function 1 INC	1/4
V390x0005.1	Active machine function 10 INC	1/4
V390x0005.2	Active machine function 100 INC	1/4
V390x0005.3	Active machine function 1000 INC	1/4
V390x0005.6	Active machine function continuous	1/4
V390x1002.0	Lubrication pulse	1/14
V39032000.0 to .2	Spindle: Set gear stage A to C	1/9
V39032000.3	Spindle: Change gear stage	1/9
V39032001.0	Spindle: Speed limit exceeded	1/9
V39032001.1	Spindle: Set speed limited	1/9
V39032001.2	Spindle: Set speed increased	1/9
V39032001.5	Spindle: Spindle in set range	1/9

Axis/spindle (NCK –> PLC)		
V39032001.7	Spindle: Actual direction of rotation CW	1/9
V39032002.3	Spindle: Tapping without compensating chuck	1/9
V39032002.5	Spindle: Active spindle mode Positioning Mode	1/9
V39032002.6	Spindle: Active spindle mode Oscillation Mode	1/9
V39032002.7	Spindle: Active spindle mode Control Mode	1/9
V390x5000.0	Stepper motor: Speed monitoring error	1/14
VD570x0000	Axis actual values (data format: FLOAT)	1/14
VD570x0004	Distances to go by the axes (data format: FLOAT)	1/14
MMC (MMC –> PLC)		
V17000000.5	M01 selected	1/5
V17000000.6	Dry run feed selected	1/11
V17000001.3	Feed override for rapid traverse selected	1/11
V17000001.7	Program test selected	1/5
V17000002.0	Select Skip Block	1/5
V18000001.0	Machine funktion: TEACH IN	1/5
V19001003.0 to .1	Axis number for handwheel 1 (A to B)	1/4
V19001003.7	Ma chine axis (is axis number for handwheel 1)	1/4
V19001004.0 to .1	Axis number for handwheel 2 (A to B)	1/4
V19001004.7	Machine axis (is axis number for handwheel 2)	1/4
MMC (PLC –> MMC)		
V19005000.2	Key lock	1/14
Machine control panel (MCP) (MCP –> PLC)		
V10000000.0 to .5	Key: T1 to T6 – free	2/
V10000000.6	Key: T7 –INC	2/
V10000000.7	Key: T8 –JOG	2/
V10000001.0	Key: T9 –REF	2/
V10000001.1	Key: T10 –AUTO	2/
V10000001.2	Key: T11 –SBL	2/
V10000001.3	Key: T12 –MDA	2/
V10000001.4	Key: T13 – spindle start +	2/
V10000001.5	Key: T14 – spindle stop	2/
V10000001.7	Key: T15 – spindle start –	2/
V10000001.2	Key: T16 –free	2/
V10000002.0	Key: T17 –free (preferably axis key)	2/
V10000002.1	Key: T18 –free (preferably axis key)	2/
V10000002.2	Key: T19 –free (preferably axis key)	2/
V10000002.3	Key: T20 –free (preferably rapid traverse, axis key)	2/
V10000002.4	Key: T21 –free (preferably axis key)	2/
V10000002.5	Key: T22 –free (preferably axis key)	2/

Machine control panel (MCP) (MCP –> PLC)		
V10000002.6	Key: T23 –free (preferably axis key)	2/
V10000002.7	Key: T24 –free	2/
V10000003.0	Key: T25 –NC RESET	2/
V10000003.1	Key: T26 –NC STOP	2/
V10000003.2	Key: T27 –NC Start	2/
VB10000004	Feed override (override value)	1/11
VB10000005	Spindle override (override value)	1/11
Machine control panel (MCP) (PLC –> MCP)		
V11000000.0 to .5	LED: L1 to L6	2/
PLC machine data		
VW45000000	Int value 1 corresponding to MD USER_DATA_INT	2/
VW45000002	Int value 2 corresponding to MD USER_DATA_INT	2/
...	Int value ... corresponding to MD USER_DATA_INT	2/
VW45000062	Int value 32 corresponding to MD USER_DATA_INT	2/
VB45001000	Hex value 1 corresponding to MD USER_DATA_HEX	2/
VB45001001	Hex value 2 corresponding to MD USER_DATA_HEX	2/
...	Hex value ... corresponding to MD USER_DATA_HEX	2/
VB45001031	Hex value 32 corresponding to MD USER_DATA_HEX	2/
VD45002000	Float value 1 corresponding to MD USER_DATA_FLOAT (4 bytes)	2/
VD45002004	Float value 2 corresponding to MD USER_DATA_FLOAT (4 bytes)	2/
...	Float value ... corresponding to MD USER_DATA_FLOAT (4 bytes)	2/
VD45002028	Float value 8 corresponding to MD USER_DATA_FLOAT (4 bytes)	2/
VB45003000	Alarm response/deletion criterion Alarm 700000 corresponding to MD USER_DATA_PLC_ALARM	2/
VB45003001	Alarm response/deletion criterion Alarm 700001 corresponding to MD USER_DATA_PLC_ALARM	2/
...	Alarm response/deletion criterion Alarm 70000... corresponding to MD USER_DATA_PLC_ALARM	2/
VB45003031	Alarm response/deletion criterion Alarm 7000031 corresponding to MD USER_DATA_PLC_ALARM	2/
User alarm (PLC –> MMC)		
V16000000.0 to V16000003.7	Activation of alarm No. 700000 to No. 700031	2/
V16001000	Variable for alarm 700000	2/
V16001004	Variable for alarm 700001	2/
...	Variable for alarm ...	2/
V16001124	Variable for alarm 700031	2/
V16002000.0	Active alarm response: NC Start inhibited	2/
V16002000.1	Active alarm response: Read–in disable	2/
V16002000.2	Active alarm response: Feed lock of all axes	2/
V16002000.3	Active alarm response: EMERGENCY STOP	2/
V16002000.4	Active alarm response: PLC STOP	2/

Index

A

Acceleration, 4-59
Access rights, 14-226
Action Single Block type, 5-90
actual values, 1-16
Actual–value acquisition, 3-42
Actual–Value Processing, 3-43
Actual–value resolution, 3-45
AUTOMATIC, 5-84
Auxiliary Function Groups, 10-188
auxiliary function output, 2-36
Auxiliary Functions, Block change, 10-187
Axis Monitoring, 1-11
    Actual Velocity, 1-15
    clamping, 1-14
    Contour, 1-12
    Positioning Monitoring, 1-13
    Zero–Speed, 1-14
Axis stop, axis stop fine, 1-14

B

Backlash Compensation, 6-116
BAG
    ready for operation, 5-85
    Stop, 5-84
block change moment, 2-36
Block Search function, 5-92

C

Channel, 5-83
Channel Status, 5-99
Compensation table, 6-118
Continuous–Path Control Mode, 2-36
contour violation, 1-20
Control direction, 3-43
Controlling Manual Traversing, 4-59
Converting, 3-41
Cyclic signal exchange, 14-224

D

D functions, 12-212
Decoding Single Block, 5-90
direction of rotation, 1-16
Drift, 6-121
Drift Compensation, 6-121
DRIFT_LIMIT, 6-122
Dry Run Feed, 5-91

E

Encoder Monitoring
    encoder frequency, 1-17
    zero mark, 1-17
Exact stop
    exact stop coarse, 2-34
    Exact stop fine, 1-13
    exact stop fine, 2-35
exact stop, 2-34

F

Feed override, 4-59, 11-199
feed override switch, 11-202
Feed stop, 5-99
Feed types, 11-193
feedoverride, 2-32
Feeds
    feed F, 11-193, 11-194
    Feed Lock, 11-201
    Feed override, 11-200
    Feed/Spindle Stop, 11-201
    Spindle override, 11-203

H

Handwheel
    assignment, 4-64
    connection, 4-63
    selection from MMC, 4-64
    Traversing in JOG Mode, 4-63
Hardware errors, 1-18
Hardware limit switches, 1-19, 4-66

I

Implicit exact stop, 2-36
Incremental Traversing, 4-62
Incremental traversing, 4-60
interface, 4-59
Interfaces, 14-224
Interrupting a Part Program, 5-95

J

JOG, 5-84
JOG mode, 4-58, 11-199

K

Kv factor, 1-12
Kv–Faktor, 3-46

L

Lead Error Compensation, 6-118

M

M function, 10-189
machine coordinate system (MCS), 7-126
MDA, 5-84
Metric/inch measuring system, 3-41
Mode, 4-67, 5-84
  interlocks, 5-88
  monitoring, 5-87
Mode change, 5-85
Monitoring Functions, 4-66

N

NC Start, 5-94
NC–Stop, 5-99

P

Part Program, 5-94
path feedrate, 3-40
Position control, 3-46
positioning accuracy, 3-41
Program Control, 5-97
  DRY, 5-97
  M01, 5-97
  PRT, 5-97
  ROV, 5-97
  SBL1, 5-97
  SBL2, 5-97
  SKP, 5-97
Program Status, 5-98
Program test, 5-89

R

rapid stop, 1-17
Rapid traverse override, 4-58
Rapid traverse override, 11-199
Rapidtraverse, 2-32
Rapidtraverse override, 2-32
Read–in disable, 5-99
Reset, 5-99, 13-220
RESET command, 5-96

S

Setpoint output, 3-42
Simulation axes, 3-42
Single block, 5-99
Single Block Mode, 5-90
Software limit switches, 1-19, 4-66
speed, control, 1-17
Speed setpoint adjustment, 3-43
Speed Setpoint Output, 3-43
spindle speed, 3-40
Spindle stop, 5-99
SPOS, 2-36
Standardization of physical quantities in the machine and setting data, 3-41
START command, 5-94
STOP commands, 5-95
Synchronization, 2-33

T

T function, 10-189
tacho adjustment, 3-43
Tool, **12-212**
  change, 12-212
  change with ”M06”, 12-212
  selecting, 12-212
  Tool compensation, 12-212
Transversal axes, 4-67

V

velocities, 3-40
velocity, 4-58, 4-64, 11-199
velocity and acceleration, 4-66
Velocity for zero–cycle blocks, 2-32

W

workpiece coordinate system (WCS, 7-126

SIEMENS AG
A&D MC BMS
Postfach 3180

D–91050 Erlangen

(Tel. +49 180 / 5050 – 222 [Hotline]
Fax +49 9131 / 98 – 2176 [Documentation]
Mailto: motioncontrol.docu@erlf.siemens.de)

Suggestions

Corrections

for Publication/Manual:

SINUMERIK 802S/802C

Manufacturer Documentation

From

Name

Company/dept.

Street

Zip code: City:

Telephone: /

Telefax: /

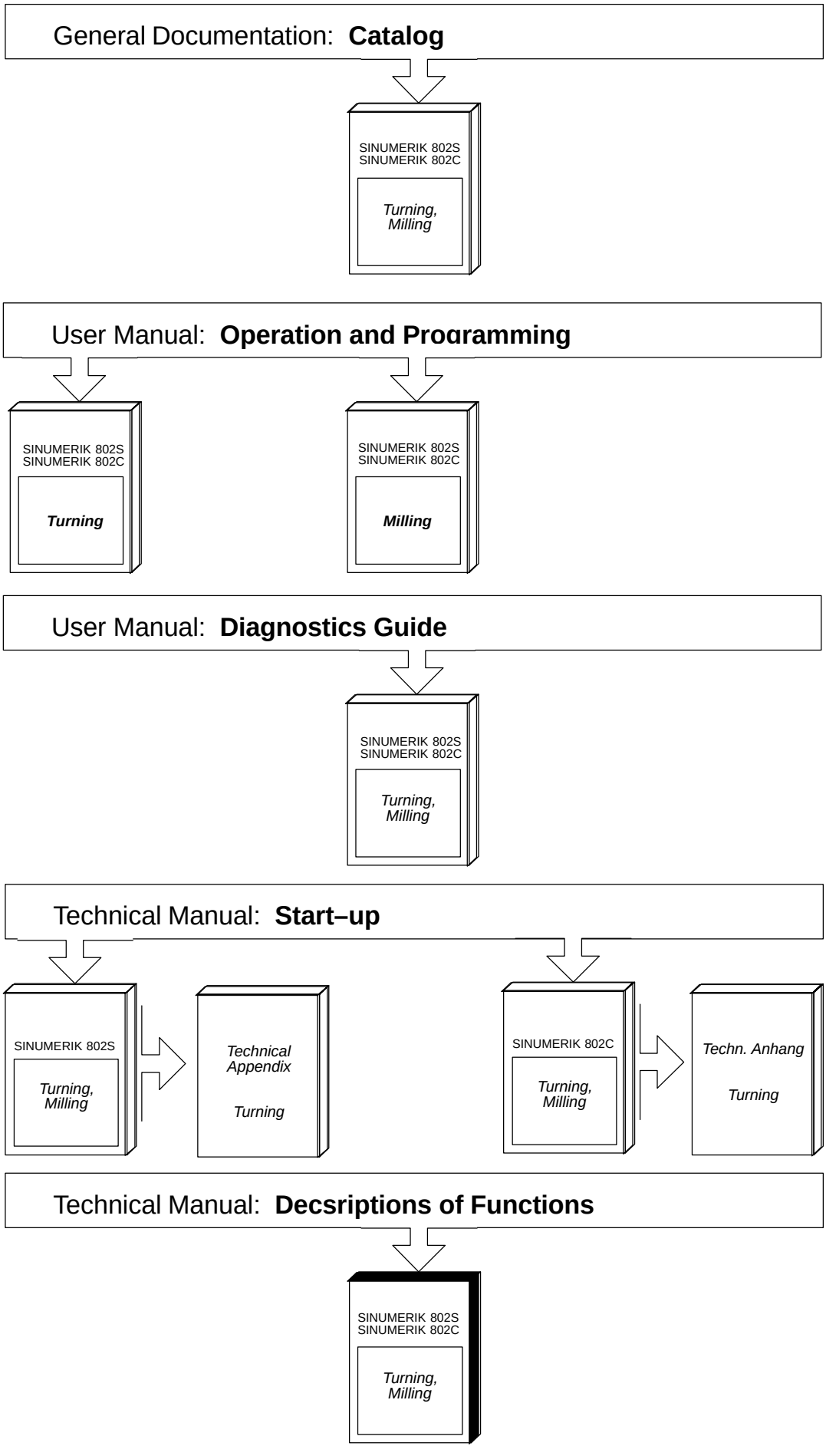
Description of Funktions

Order No.: 6FC5597–3AA10–0BP2
Edition: 01.02

Should you come across any printing errors when reading this publication, please notify us on this sheet.
Suggestions for improvement are also welcome.

Suggestions and/or corrections

Document Structure SINUMERIK 802S and 802C



Siemens AG
Automatisierungs- und Antriebstechnik
Motion Control Systems
Postfach 3180, D – 91050 Erlangen
Bundesrepublik Deutschland

www.ad.siemens.de

© Siemens AG 2002
Subject to change without prior notice
Order No.: 6FC5597-3AA10-0BP2

Printed in the Federal Republic of Germany