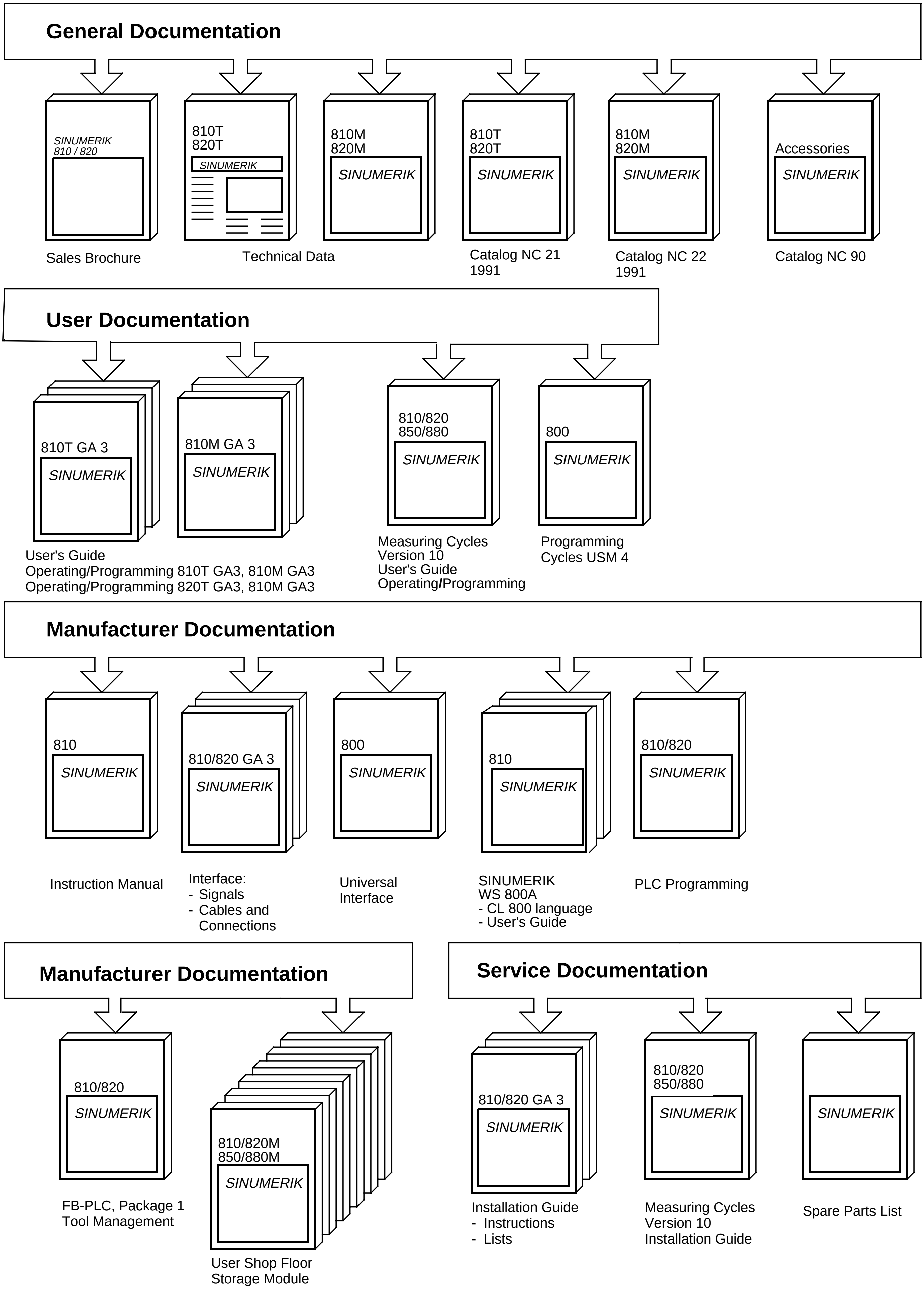
SINUMERIK 810 GA3
SINUMERIK 820 GA3
Software Version 3
Installation Instructions

Installation Guide 01.93 Edition

Service Documentation

SINUMERIK 810/820, Basic Version 3 (GA 3)



SINUMERIK 810 GA3
SINUMERIK 820 GA3
Software Version 3
Installation Instructions

Service Documentation

Installation Guide

Applies to:

Control		*Software Version*
SINUMERIK 810M	Basic Version 3	3
SINUMERIK 810T	Basic Version 3	3
SINUMERIK 820M	Basic Version 3	3
SINUMERIK 820T	Basic Version 3	3

January 1993 Edition

SINUMERIK® documentation

Printing history

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

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

Status code in ”Remarks” column:

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

Edition	**Order No.**	**Remarks**
11.90	6ZB5 410-0DL02-0AA0	**A**
09.91	6ZB5 410-0DL02-0AA1	**C**
01.93	6ZB5 410-0DL02-0AA2	**C**

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.

This publicaation was produced on the Siemens 5800 Office System.
Subject to change without prior notice.

Preliminary remarks

Notes for the reader

This documentation is intended for specialists involved in the installation and servicing of the SINUMERIK 810, Basic Version 3 (brief designation: GA3) and SINUMERIK 820, Basic Version 3.

The **Installation Guide** for the **SINUMERIK 810 GA3/820 GA3** is broken down into two parts:

- Installation Instructions
- Installation Lists

Contents

1 Preconditions and Visual Inspection

1.1 Preconditions for installation

- Electrical and mechanical installation of the machine must have been completed and the axes prepared for operation (to be confirmed by the customer).
- Customer PLC program operational and pretested.
- Measuring system installed and wired as far as SINUMERIK (visual check).

  Applies only with analog link to drive.

  If the customer has fitted **adapter plugs** in the measuring-circuit leads:
  Check for proper connection, strain relief and above all the prescribed shielding.
- Cables connected to the machine. Cable shields run to the control neutral point as per Interface Description.

  Flexible **earth wires** installed (visual check):
  – Earth bar in SINUMERIK interface unit 10 mm^2
  – Earth bar in machine column interface unit 10 mm^2
  – All earthing points on the equipment are connected
- Customer personnel support for work on the interface unit, machine operation and customer-produced PLC program.

  Recommendation:

  Limit travel ranges (greater clearance distances) by displacing the end stop (EMERGENCY STOP cam).

- The specified machine data must be available.
- Test tapes must be available for checking machine-specific functions.

1.2 Preliminary remarks

Synthetic or rubber soiling, and in particular synthetic flooring and carpeting, may produce static charging of several kilovolts in human beings. Integrated circuits are sensitive as far as high-voltage discharge of this type is concerned. Consequently, never touch the printed conductors or components without first discharging on an earthed system part.

Modules and power supply cables should only be withdrawn or inserted with the control disconnected.

Even with the control disconnected, attention should be paid to static charging to prevent short-circuiting across the VCC RAM printed conductors, otherwise data in the buffered CMOS RAM memories may be corrupted or printed conductors may even burn out.

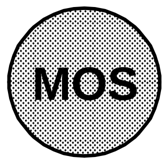
MOS Caution! Observe safety regulations!

MOS Caution! Observe safety regulations!

MOS is a technology used to produce integrated digital circuits. ”MOS” stands for Metal Oxide Silicon. Its principal advantages are:

- Simple transistor design
- High component density
- Extremely low power consumption

Identification on packaging



Identification on printed circuit board

M
O
S

Caution!

The printed circuit board is fitted with MOS chips.

To prevent these chips from being irreparably damaged, equipotential bonding must be ensured prior to installing the PCB.

Remove the PCB together with the conductive foam plastic from the packaging and touch an earthed system part.

Do not touch the printed conductors and components!

Note in packaging.

Additional notes:

Do not open the special packaging unnecessarily.

Store only in the black (conductive) foam plastic.

Do not bring into contact with plastic materials (possibility of static charging).

Disconnect power supply prior to installation and removal.

1.3 Visual inspection of system

1.3.1 Earthing

Proper earthing to divert external malfunctions is vital for trouble-free operation. It must be ensured that the earth wires are not kinked and have the necessary cross-section (see also Instruction Manual).

1.3.2 Position encoders

Particular attention is to be paid to the prescribed installation of the graduated scales (air gap etc.) and the pulse generators (coupling), see also Heidenhain Installation and Adjustment Instructions. Check for correct wiring and fixed location of the connectors.
Other makes of position encoder may result in inaccuracy and surface quality problems.

1.3.3 Cable laying

It should be ensured as far as possible that power cables and control cables are isolated. Do not produce earth loops since such loops or non-regulation earthing may generate a ripple voltage affecting the speed controller setpoint. Smooth running at minimum speeds is then no longer guaranteed.

- Is there any kinking (fibre-optics cables)?
- Proper running of cables? Cable dragging?

1.4 Shielding

The overall shields of all cables running to or from the control must be earthed at the control via the connectors (see Interface Description Part 2, Section 4 ”Voltage and Functional Tests/Drift”).

1.5 Operator panel

Check that the pushbuttons, keys, lamps, symbols and screen are in order.

1.6 Overall condition

Check the module mounts and cover plates. Check whether the front panel screws have been tightened (M connection).

Accessory pack:
Log book and complete parts list available?
(The parts list is included with the original delivery note and must be inserted in the log book).

When replacing modules or in the event of a fault, check all ICs in sockets for correct location and fit.

Check jumper assignment on modules.

1.6.1 Battery adjacent to power supply unit

The battery holder is located on the right-hand side of the monitor housing. The battery should only be replaced under power so that the stored data is not lost. The battery voltage is continuously monitored. Alarm 1 is displayed if the voltage drops below 2.7V.

Battery type:	3 x 1.5 V (mignon cell, LR 6)
Backup period:	Approx. 1 year
Note:	The power supply can be unplugged while the device is not under power without data being lost. Only change the battery while the control is under power.

1.6.2 Battery backup of the RAM memory

While the control is not under power the RAM memory on the CPU must be backed up by a battery. The battery next to the power supply unit backs up the RAM. In this way the following data remain stored:

- PLC user program
- User texts (PLC alarm texts and PLC message texts)
- Machine data
- Part programs
- Tool offsets
- R parameters
- Zero offsets
- Setting data
- Real-time clock

1.6.3 Cables

Check all cables in accordance with the cable and equipment overview (in Interface Description 2). This applies particularly to cables produced by the customer. **A random check should be made on at least one connector** (particular attention must be paid to conductive elastomer connections!). In the event of failure to comply with our guidelines, the responsible sales point must be advised and any necessary corrective measures instigated.

Ensure that any fibre-optics cables are properly installed (bend radius as prescribed in installation specifications).

2 Installation Checklist

810 (GA3)		☐
820 (GA3)		☐
Cycles used		
T	M	
☐	☐	

Serial No.	________________
Software Version	________________

Installation sequence

Section 1 of the Installation Guide must be observed!

Copy installation checklist.

Mark Yes/No as applicable after each section has been completed.

Enter all required values where stated.

Complete the NC machine data list. PLC machine data list and setting data list after installation.

	First installation	Second installation
Name		
Date		
Office		
	Manufacturer:	Customer:
Address		

Explanations of the following points are to be found in the individual sections of this guide.

1.	Are the preconditions for installation fulfilled as described in Section 1?	yes	no
2.	Visual inspection: Supply connection, EMERGENCY STOP, earthing, position encoders, cable laying, shielding, keyboard, overall condition ok?	yes	no
3.	Voltage test performed as described in Section 4?	yes	no
4.	Standard installation completed and the customer specific machine data entered?	yes	no
5.	PLC program entered and tested (safety functions)?	yes	no
6.	Position control loops of axes installed and the following checked: Axis speeds / tacho generator compensation / multiplication factors / servo gain (K_V factor) / acceleration / exact stop / position control loop monitors / analog spindle speed / traversing ranges?	yes	no
7.	Have all JOG functions been tested?	yes	no
	Has the function check with the test program been performed by the customer?	yes	no
8.	Was a data backup made of the following data:		
	• NC machine data, PLC machine data	yes	no
	• PLC message and alarm texts	yes	no
	• Zero offsets	yes	no
	• Setting data	yes	no
	• R parameters	yes	no
	• Tool offsets	yes	no
9.	Was this data placed at the machine?	yes	no
10.	Installation checklist filled in completely (NC MD, PLC MD, setting data, options), put with the log book and placed at the control?	yes	no
11.	Were the following functions explained to the customer: drift compensation, reference point setting, backlash compensation?	yes	no
12.	Has the installation protocol been signed by the customer?	yes	no

<u>Signatures</u>

1st installation **2nd installation**

----------------------------------- **---------------------------------**

NC machine data list

Units: See detailed description of machine data.

This list is to be completed following commissioning and attached to the log book with the punched tape.

No.	Value	Designation
1		Speed behind pre-lim. sw.
3		Corner deceleration rate.
5		No. of MIB parameters
6		Threshold for CRC insertion
7		Circle end pos. monitoring.
8		Max. number of PPs>s
9		Error window repos.
10		Feed after block search
11		ID word
13		Number of TO parameters
14		Cycles MD (from)
15		Cycles MD (to)
16		Cycles SD (from)
17		Cycles SD (to)
20		Large circular buffer
100/		Positions of
121		feedrate override switch
131/		Positions of
146		spindle override switch
147/		Positions of
154		rapid traverse override switch
155		Sampling time position controller
156		Cutout delay servo enable
157		Control type software version display
200/223		Keypad assignments SIN. 810
225		Threshold value circle center offset
250		Language selection
260		Selection rotary axis mode
261		Deselection rotary axis mode
1080		Reset pos. for G group 0 channel 1
1081		Reset pos. for G group 0 channel 2
1100		Reset pos. for G group 2 channel 1
1101		Reset pos. for G group 2 channel 2
1120		Reset pos. for G group 5 channel 1
1121		Reset pos. for G group 5 channel 2
1140		Reset pos. for G group 7 channel 1
1141		Reset pos. for G group 7 channel 2
1180		Reset pos. for G group 11 channel 1
1181		Reset pos. for G group 11 channel 2
2000		Axis assignment 1st axis
2001		Axis assignment 2nd axis
2002		Axis assignment 3rd axis
2003		Axis assignment 4th axis
2004		Axis assignment 5th axis
2005		Axis assignment 6th axis
2006		Axis assignment 7th axis

No.	Value	Designation
2040		Exact stop course 1st axis
2041		Exact stop course 2nd axis
2042		Exact stop course 3rd axis
2043		Exact stop course 4th axis
2044		Exact stop course 5th axis
2045		Exact stop course 6th axis
2046		Exact stop course 7th axis
2080		Exact stop fine 1st axis
2081		Exact stop fine 2nd axis
2082		Exact stop fine 3rd axis
2083		Exact stop fine 4th axis
2084		Exact stop fine 5th axis
2085		Exact stop fine 6th axis
2086		Exact stop fine 7th axis
2120		Zero speed control
2121		Zero speed control
2122		Zero speed control
2123		Zero speed control
2124		Zero speed control
2125		Zero speed control
2126		Zero speed control
2160		No effect
2161		No effect
2162		No effect
2163		No effect
2164		No effect
2165		No effect
2166		No effect
2200		Backlash compensation
2201		Backlash compensation
2202		Backlash compensation
2203		Backlash compensation
2204		Backlash compensation
2205		Backlash compensation
2206		Backlash compensation
2240		Software limit switch 1 pos
2241		Software limit switch 1 pos
2242		Software limit switch 1 pos
2243		Software limit switch 1 pos
2244		Software limit switch 1 pos
2245		Software limit switch 1 pos
2246		Software limit switch 1 pos
2280		Software limit switch 1 neg
2281		Software limit switch 1 neg
2282		Software limit switch 1 neg

No.	Value	Designation
2283		Software limit switch 1 neg
2284		Software limit switch 1 neg
2285		Software limit switch 1 neg
2286		Software limit switch 1 neg
2320		Software limit switch 2 pos
2321		Software limit switch 2 pos
2322		Software limit switch 2 pos
2323		Software limit switch 2 pos
2324		Software limit switch 2 pos
2325		Software limit switch 2 pos
2326		Software limit switch 2 pos
2360		Software limit switch 2 neg
2361		Software limit switch 2 neg
2362		Software limit switch 2 neg
2363		Software limit switch 2 neg
2364		Software limit switch 2 neg
2365		Software limit switch 2 neg
2366		Software limit switch 2 neg
2400		Reference point value
2401		Reference point value
2402		Reference point value
2403		Reference point value
2404		Reference point value
2405		Reference point value
2406		Reference point value
2440		Reference point shift
2441		Reference point shift
2442		Reference point shift
2443		Reference point shift
2444		Reference point shift
2445		Reference point shift
2446		Reference point shift
2480		Tool reference value
2481		Tool reference value
2482		Tool reference value
2483		Tool reference value
2484		Tool reference value
2485		Tool reference value
2486		Tool reference value
2520		Kv factor
2521		Kv factor
2522		Kv factor
2523		Kv factor
2524		Kv factor

No.	Value	Designation
2525		Kv factor
2526		Kv factor
2560		Difference time constant
2561		Difference time constant
2562		Difference time constant
2563		Difference time constant
2564		Difference time constant
2565		Difference time constant
2566		Difference time constant
2600		Multgain
2601		Multgain
2602		Multgain
2603		Multgain
2604		Multgain
2605		Multgain
2606		Multgain
2640		Drive error threshold
2641		Drive error threshold
2642		Drive error threshold
2643		Drive error threshold
2644		Drive error threshold
2645		Drive error threshold
2646		Drive error threshold
2680		Max. command speed
2681		Max. command speed
2682		Max. command speed
2683		Max. command speed
2684		Max. command speed
2685		Max. command speed
2686		Max. command speed
2720		Drift compensation
2721		Drift compensation
2722		Drift compensation
2723		Drift compensation
2724		Drift compensation
2725		Drift compensation
2726		Drift compensation
2760		Acceleration
2761		Acceleration
2762		Acceleration
2763		Acceleration
2764		Acceleration
2765		Acceleration
2766		Acceleration

No.	Value	Designation
2800		Max. speed
2801		Max. speed
2802		Max. speed
2803		Max. speed
2804		Max. speed
2805		Max. speed
2806		Max. speed
2840		Ref. point cutoff speed
2841		Ref. point cutoff speed
2842		Ref. point cutoff speed
2843		Ref. point cutoff speed
2844		Ref. point cutoff speed
2845		Ref. point cutoff speed
2846		Ref. point cutoff speed
2880		Jog feedrate
2881		Jog feedrate
2882		Jog feedrate
2883		Jog feedrate
2884		Jog feedrate
2885		Jog feedrate
2886		Jog feedrate
2920		Jog rapid traverrse
2921		Jog rapid traverrse
2922		Jog rapid traverrse
2923		Jog rapid traverrse
2924		Jog rapid traverrse
2925		Jog rapid traverrse
2926		Jog rapid traverrse
2960		Ref. point approach speed
2961		Ref. point approach speed
2962		Ref. point approach speed
2963		Ref. point approach speed
2964		Ref. point approach speed
2965		Ref. point approach speed
2966		Ref. point approach speed
3000		Incremental feedrate
3001		Incremental feedrate
3002		Incremental feedrate
3003		Incremental feedrate
3004		Incremental feedrate
3005		Incremental feedrate
3006		Incremental feedrate
3040		Interpolation parameter
3041		Interpolation parameter
3042		Interpolation parameter
3043		Interpolation parameter
3044		Interpolation parameter
3045		Interpolation parameter
3046		Interpolation parameter

No.	Value	Designation
3160		Pointer compensation pos.
3161		Pointer compensation pos.
3162		Pointer compensation pos.
3163		Pointer compensation pos.
3164		Pointer compensation pos.
3165		Pointer compensation pos.
3166		Pointer compensation pos.
3200		Pointer compensation neg.
3201		Pointer compensation neg.
3202		Pointer compensation neg.
3203		Pointer compensation neg.
3204		Pointer compensation neg.
3205		Pointer compensation neg.
3206		Pointer compensation neg.
3240		Distance between 2 values
3241		Distance between 2 values
3242		Distance between 2 values
3243		Distance between 2 values
3244		Distance between 2 values
3245		Distance between 2 values
3246		Distance between 2 values
3280		Compensation value
3281		Compensation value
3282		Compensation value
3283		Compensation value
3284		Compensation value
3285		Compensation value
3286		Compensation value
3320		Contour monit. tol. band
3321		Contour monit. tol. band
3322		Contour monit. tol. band
3323		Contour monit. tol. band
3324		Contour monit. tol. band
3325		Contour monit. tol. band
3326		Contour monit. tol. band
3360		Threshold contour speed
3361		Threshold contour speed
3362		Threshold contour speed
3363		Threshold contour speed
3364		Threshold contour speed
3365		Threshold contour speed
3366		Threshold contour speed
3400		Tool change position
3401		Tool change position
3402		Tool change position
3403		Tool change position
3404		Tool change position
3405		Tool change position
3406		Tool change position

No.	Value	Designation	No.	Value	Designation
3440		Modulo f. endless rot. axis with LEC*	3763		Prelimit switch
3441		Modulo f. endless rot. axis with LEC*	3764		Prelimit switch
3442		Modulo f. endless rot. axis with LEC*	3765		Prelimit switch
3443		Modulo f. endless rot. axis with LEC*	3766		Prelimit switch
3444		Modulo f. endless rot. axis with LEC*	3800		2nd Kv factor for G36
3445		Modulo f. endless rot. axis with LEC*	3801		2nd Kv factor for G36
3446		Modulo f. endless rot. axis with LEC*	3802		2nd Kv factor for G36
3480		Software Exe	3803		2nd Kv factor for G36
3481		Software Exe	3804		2nd Kv factor for G36
3482		Software Exe	3805		2nd Kv factor for G36
3483		Software Exe	3806		2nd Kv factor for G36
3484		Software Exe	3840		Grad. period of linear scale
3485		Software Exe	3841		Grad. period of linear scale
3486		Software Exe	3842		Grad. period of linear scale
3520		2nd Kv factor	3843		Grad. period of linear scale
3521		2nd Kv factor	3844		Grad. period of linear scale
3522		2nd Kv factor	3845		Grad. period of linear scale
3523		2nd Kv factor	3846		Grad. period of linear scale
3524		2nd Kv factor	3880		Weighting factor
3525		2nd Kv factor	3881		Weighting factor
3526		2nd Kv factor	3882		Weighting factor
3600		Sym. time constant	3883		Weighting factor
3601		Sym. time constant	3884		Weighting factor
3602		Sym. time constant	3885		Weighting factor
3603		Sym. time constant	3886		Weighting factor
3604		Sym. time constant	3920		Absolute offset
3605		Sym. time constant	3921		Absolute offset
3606		Sym. time constant	3922		Absolute offset
3640		No. pulses var. incr. weighting	3923		Absolute offset
3641		No. pulses var. incr. weighting	3924		Absolute offset
3642		No. pulses var. incr. weighting	3925		Absolute offset
3643		No. pulses var. incr. weighting	3926		Absolute offset
3644		No. pulses var. incr. weighting	4000		Spindle assignment
3645		No. pulses var. incr. weighting	4001		Spindle assignment
3646		No. pulses var. incr. weighting	4010		Drift compensation
3680		Travel for var. incr. weighting	4011		Drift compensation
3681		Travel for var. incr. weighting	4020		Zero mark offset
3682		Travel for var. incr. weighting	4021		Zero mark offset
3683		Travel for var. incr. weighting	4030		Max. speed gear 1
3684		Travel for var. incr. weighting	4031		Max. speed gear 1
3685		Travel for var. incr. weighting	4040		Max. speed gear 2
3686		Travel for var. incr. weighting	4041		Max. speed gear 2
3720		Delay zero speed control	4050		Max. speed gear 3
3721		Delay zero speed control	4051		Max. speed gear 3
3722		Delay zero speed control	4060		Max. speed gear 4
3723		Delay zero speed control	4061		Max. speed gear 4
3724		Delay zero speed control	4070		Max. speed gear 5
3725		Delay zero speed control	4071		Max. speed gear 5
3726		Delay zero speed control	4080		Max. speed gear 6
3760		Prelimit switch	4081		Max. speed gear 6
3761		Prelimit switch	4090		Max. speed gear 7
3762		Prelimit switch	4091		Max. speed gear 7

* LEC = leadscrew error compensation

No.	Value	Designation
4100		Max. speed gear 8
4101		Max. speed gear 8
4110		Min. speed gear 1
4111		Min. speed gear 1
4120		Min. speed gear 2
4121		Min. speed gear 2
4130		Min. speed gear 3
4131		Min. speed gear 3
4140		Min. speed gear 4
4141		Min. speed gear 4
4150		Min. speed gear 5
4151		Min. speed gear 5
4160		Min. speed gear 6
4161		Min. speed gear 6
4170		Min. speed gear 7
4171		Min. speed gear 7
4180		Min. speed gear 8
4181		Min. speed gear 8
4190		Accel. time const gear 1
4191		Accel. time const gear 1
4200		Accel. time const gear 2
4201		Accel. time const gear 2
4210		Accel. time const gear 3
4211		Accel. time const gear 3
4220		Accel. time const gear 4
4221		Accel. time const gear 4
4230		Accel. time const gear 5
4231		Accel. time const gear 5
4240		Accel. time const gear 6
4241		Accel. time const gear 6
4250		Accel. time const gear 7
4251		Accel. time const gear 7
4260		Accel. time const gear 8
4261		Accel. time const gear 8
4270		Cutoff speed M19 gear 1
4271		Cutoff speed M19 gear 1
4280		Cutoff speed M19 gear 2
4281		Cutoff speed M19 gear 2
4290		Cutoff speed M19 gear 3
4291		Cutoff speed M19 gear 3
4300		Cutoff speed M19 gear 4
4301		Cutoff speed M19 gear 4
4310		Cutoff speed M19 gear 5
4311		Cutoff speed M19 gear 5
4320		Cutoff speed M19 gear 6
4321		Cutoff speed M19 gear 6
4330		Cutoff speed M19 gear 7
4331		Cutoff speed M19 gear 7

No.	Value	Designation
4340		Cutoff speed M19 gear 8
4341		Cutoff speed M19 gear 8
4350		Gain M19 gear 1
4351		Gain M19 gear 1
4360		Gain M19 gear 2
4361		Gain M19 gear 2
4370		Gain M19 gear 3
4371		Gain M19 gear 3
4380		Gain M19 gear 4
4381		Gain M19 gear 4
4390		Gain M19 gear 5
4391		Gain M19 gear 5
4400		Gain M19 gear 6
4401		Gain M19 gear 6
4410		Gain M19 gear 7
4411		Gain M19 gear 7
4420		Gain M19 gear 8
4421		Gain M19 gear 8
4430		Positional tolerance M19
4431		Positional tolerance M19
4440		Spindle speed tolerance
4441		Spindle speed tolerance
4450		Max. spindle speed tolerance
4451		Max. spindle speed tolerance
4460		Zero-speed tolerance
4461		Zero-speed tolerance
4470		Delay servo enable
4471		Delay servo enable
4480		Minimum motor speed
4481		Minimum motor speed
4490		Set speed
4491		Set speed
4500		Reciprocation speed
4501		Reciprocation speed
4510		Max. spindle speed
4511		Max. spindle speed
4520		Spindle pos. ext. M19
4521		Spindle pos. ext. M19
4590		No. pulses spindle encoder
4591		No. pulses spindle encoder
10960		Fictitious 2nd ref. point
10961		Fictitious 2nd ref. point
10962		Fictitious 2nd ref. point
10963		Fictitious 2nd ref. point
10964		Fictitious 2nd ref. point
10965		Fictitious 2nd ref. point
10966		Fictitious 2nd ref. point

No.	Bits 7	6	5	4	3	2	1	0
5000	0	0	0	0				
5001	0	0	0	0				
5002								
5003								
5004								
5005								
5006								
5007								
5008								
5009								
5010								
5011								
5012								
5013								
5014								
5015								
5016								
5018								
5040								
5041								
5042								
5061								
5062								
5063								
5064								
5065								
5066								
5200								
5201		0	0	0	0	0		0
5210								
5211								
5400								
5401								
5460								
5461								
5480								
5481								

No.	Bits 7	6	5	4	3	2	1	0
5500								
5501								
5520								
5521								
5540								
5541								
5580								
5581								
5600								
5601								
5602								
5603								
5604								
5605								
5606								
5640								
5641								
5642								
5643								
5644								
5645								
5646								
5680								
5681								
5682								
5683								
5684								
5685								
5686								
5720								
5721								
5722								
5723								
5724								
5725								
5726								

	Bits							
No.	7	6	5	4	3	2	1	0
5760								
5761								
5762								
5763								
5764								
5765								
5766								
5800								
5801								
5802								
5803								
5804								
5805								
5806								

	Bits							
No.	7	6	5	4	3	2	1	0
5840								
5841								
5842								
5843								
5844								
5845								
5846								

The compensation flags for leadscrew error compensation (MD 6000 - 6249) must, if necessary, be written on a separate sheet.

PLC machine data list

No.	Value	Designation	Units
0		**Interrupt input byte no.**	
1		**Max. interpreter runtime 0B1+0B2**	%
2		**PLC call OB1**	IPO clock
3		**Max. interpr. 0B2**	µs
4			
5		**Scan time monitor**	ms
6		**No. of last S5 time**	
7			
8		**No. of interf. for DB37**	–
9		**Selection Kbyte for DB area**	
10			
1000		**1st PLC user MD**	
1001		**2nd PLC user MD**	
1002		**3rd PLC user MD**	
1003		**4th PLC user MD**	
1004		**5th PLC user MD**	
1005		**6th PLC user MD**	
1006		**7th PLC user MD**	
1007		**8th PLC user MD**	

	Bits							
No.	7	6	5	4	3	2	1	0
2000								
2001					0	0	0	0
2002								
2003								
3000								
3001								
3002								
3003								

Setting data lists

	Bits							
No.	7	6	5	4	3	2	1	0
5000	0	0	0	0	0			
5001	0	0	0	0	0	0	0	
5010								
5011								
5012								
5013								
5014								
5015								
5016								
5017								

	Bits							
No.	7	6	5	4	3	2	1	0
5018								
5019								
5020								
5021								
5022								
5023								
5024								
5025	0	0	0	0	0	0	0	
5026								
5027								
5028								
5029								
5600								
5601								
5602								
5603								
5604								

Options list

Yes	No	Order code	Options	Remarks
		A04 A05 A21 A22 A23	4th axis (M) 5th axis (M) 1st auxiliary axis (T) 2nd auxiliary axis (T) 3rd auxiliary axis (T)	2nd meas. circuit module ” ” ” ”
		B01 B02 B03	Tape reader I. T61 without winder Tape reader I. T40 with winder Tape reader I. T40 with winder	- - -
		B21 B25	Tape reader I. T41 with small front panel Program sequencing with diskette drive	Otherwise as for T40 -
		B52 B60 B61	Interpol. tapping/threading (G36) Spline interpolation 3D/helical interpolation	
		B65	Transmit	
		B70	Polar coordinates, drilling and milling patterns	810M GA3
		B73	Cylindrical interpolation	
		B75	Blueprint programming	
		B78	Inprocess (high-speed) measurement	
		C47 C48 C49	Program memory expansion to 64 Kbytes Program memory expansion to 96 Kbytes Program memory expansion to 128 Kbytes	 No PLC memory expansion possible
		C60 C62	PLC controlled, serial interface 2nd V.24 (RS 232) interface	
		C69	Block transfer	
		E31	Thread cutting and feedrate per revolution	810M
		E36	Tapping without compensation chuck	Precondition 1 meas.-circuit and F05

Yes	No	Order code	Options	Remarks
		E42	Oriented spindle stop M19	Analog spindle speed F05 and F06
		B86	Software cam	
		B88	Axis duplication	
		F05	Analog spindle speed for spindle 1	Only for 810M
		F06	Analog spindle speed for spindle 2	
		F72	External data input (from the PLC)	
		H56	Leadscrew error compensation	
		J16	Graphics simulation	
		J23 J24 J25 J53 J54 J55 J26 J27 J28 J29 J31 J35 J36 J39	French/German Italian/German Spanish/German French/English Italian/English Spanish/English Dutch/German Russian/German Swedish/German Finnish/German Polish/German Hungarian/German Czech/German Turkish/German	
		J03	Color monitor	
		J81 J82 J82 J85	Integr. machine control panel (SINUMERIK 810 only) Integr. cust. operator panel (SINUMERIK 820 only) Rapid traverse override switch (SINUMERIK 820 only) Ext. machine control panel for 5 axes	No J82, J85 or J97 No J81 No J81, J96
		J96	External machine control panel (7 axes with rotary switch x, z, c, 4, 5, 6, 7)	No J81, J95

Yes	No	Order code	Options	Remarks
		 K20 K21 K23 K24 K25 K70 K71 K72 K75	Measuring-circuit modules EXE cannot be fitted EXE can be fitted with 1 5/10-fold EXE with 2 5/10-fold EXE with 3 5/10-fold EXE HMS module 3 actual values (width 20 mm) HMS module 3 actual values 3 setpoints (width 40 mm) HMS module 3 actual values 6 setpoints (width 40 mm) HMS module 3 actual values 3 setpoints HMS/absolute submodule for 3 axes (width 40 mm)	
		M01	I/O submodule, basic submodule with mounting plate	
		M02	I/O submod., expans. submod.	M01 or M04
		M03	I/O submod., expans. submod.	M01 or M04
		M04	I/O submodule, basic submodule (64 inputs)	
		M10	Handwheel interface submodule	M01 or M04
		M11	Handwheel interface submodule with mounting plate	
		M41	I/O submodule, basic submodule with mounting plate, w/o cable	P07 required
		M42	I/O submodule, expansion submodule (identical with M02)	M41 or M44 required
		M43	I/O submodule, expansion submodule (identical with M03)	M41 or M44 required
		M44	I/O submodule, basic submodule (64 inputs)	P07 required
		M49	Round cable 1m (for I/O sub-modules)	M01, M04, M11

Yes	No	Order code	Options	Remarks
		M50	Round cable 5m (for I/O sub-modules)	M01, M04, M11
		 M51 M52 M53 M54 M55	Ribbon cable for connecting: 1 submodule 2 submodules 3 submodules 4 submodules 5 submodules (5th submodule handwheel)	M49 bzw. M50 erforderlich
		M62 M63 M64 M65	Ribbon cable for P07+I/O basic submodule Ribbon cable for P07+I/O basic submodule+ 1 I/O expansion module Ribbon cable for P07+I/O basic submodule+ 2 I/O expansion modules Ribbon cable for P07+I/O basic submodule+ 3 I/O expansion modules	Precondition N96
		N06	Integrated auxiliary axis	Precondition P06 or P08,F72
		N23	RAM-UMS-SUBMODULE 128 Kbytes	
		N24	RAM-UMS-SUBMODULE 256 Kbytes	
		N31	Configurability+UMS-EPROM-submodule 128 Kbytes not programmed	
		N32	Configurability+UMS-EPROM-submodule 256 Kbytes not programmed	
		  N60 N61 N65 N66 N68 N70 N71 N72 N73	INPUT/OUTPUT modules for PLC expansion unit 32 inputs type K 32 inputs type U 32 inputs 0.5A type K 32 inputs 0.5A type U 16 outputs 2A type U 16 outputs 1A type K 64 inputs type SINUMERIK 32 outp. 0.5A, type SINUMERIK 32 outp. 2A, type SINUMERIK	 P06/P08 P06/P08 P06/P08 P06/P08 P06/P08 P06/P08 P06/P08 P06/P08 P06/P08
		N75	Link to SIMATIC S5 (master PLC)	

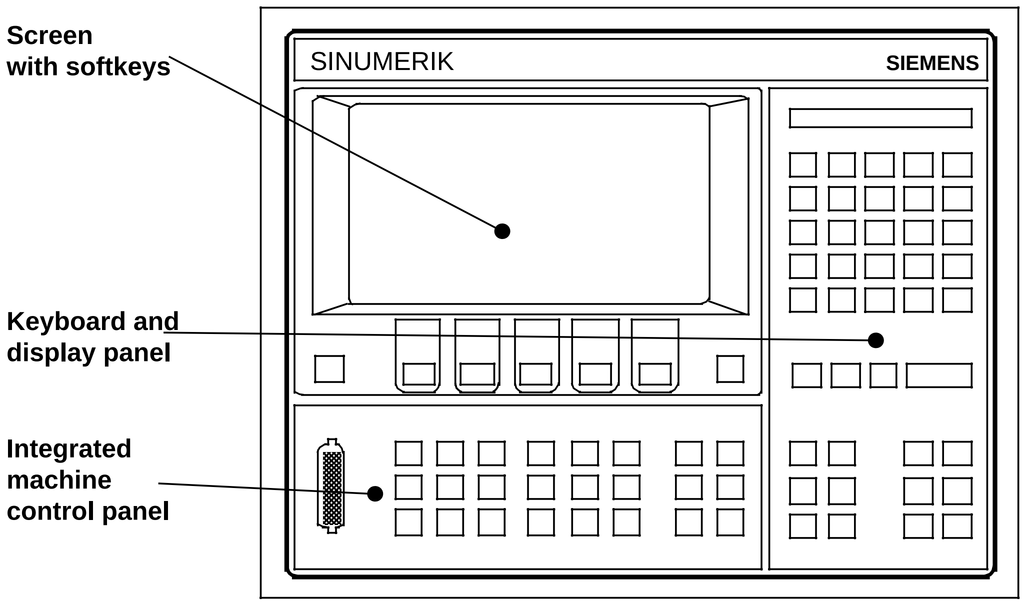
Yes	No	Order code	Options	Remarks
		N96	MPC link module	
		P06	PLC expansion unit with 10 standard slots	Excludes P07 or P08
		P07	Distributed link of I/O submodules	Excludes P06 or P08
		P08	PLC expansion unit (maxi-EU)	Prerequisite N96

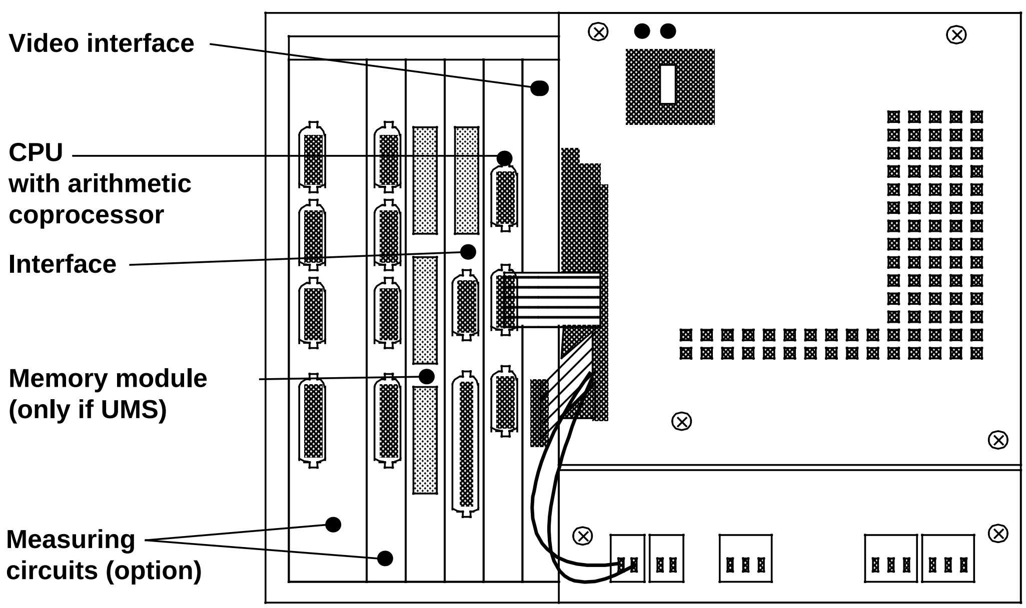
3 Overview of Modules and Standard Jumperings

3.1. Basic system

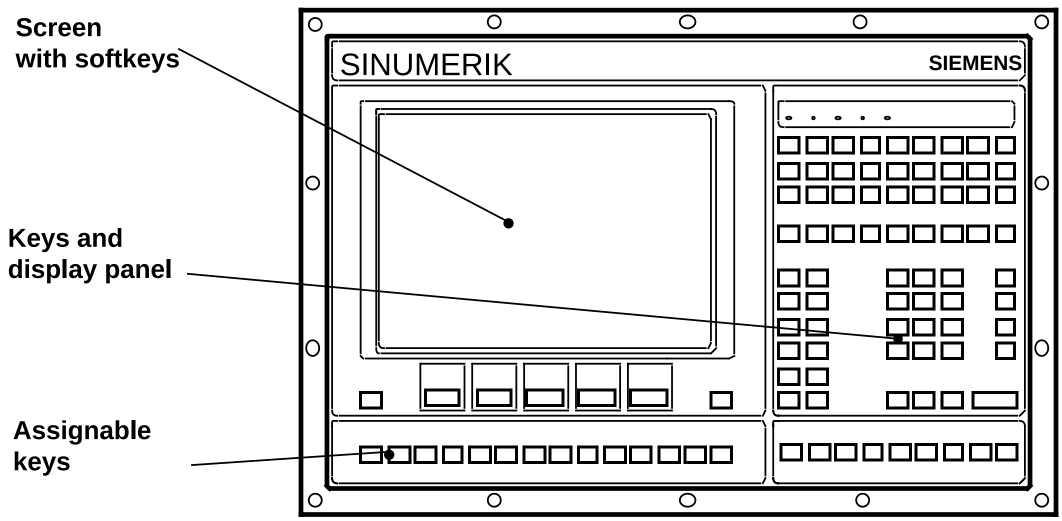
3.1.1 SINUMERIK 810 GA3

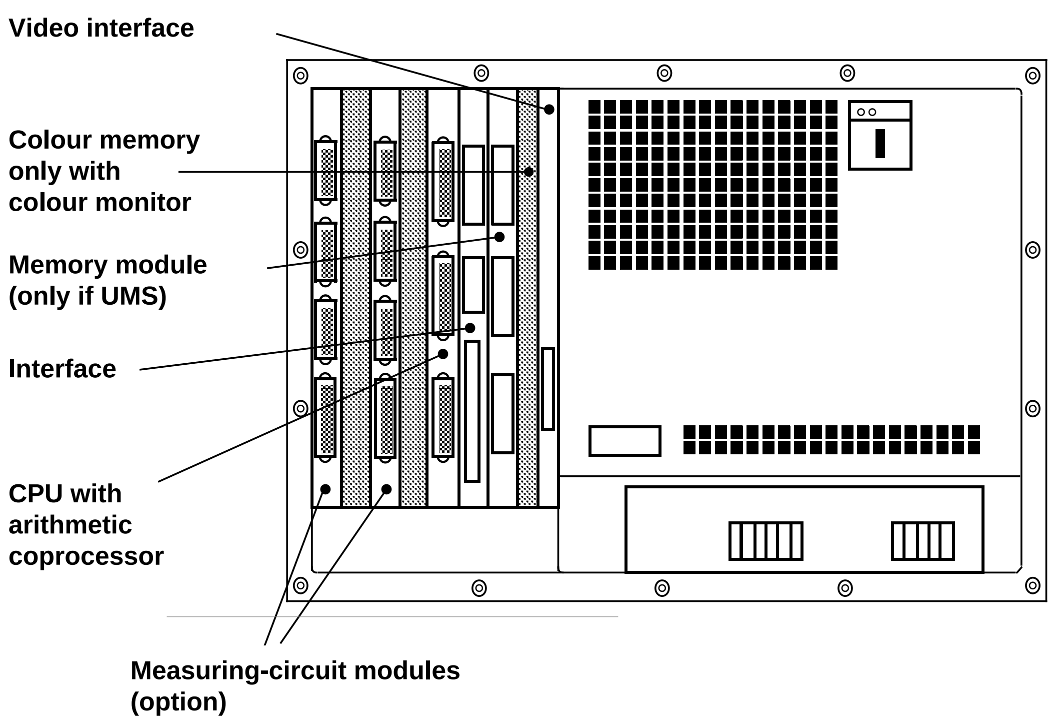
In order to save space when mounting the control on the machine and to avoid problems when linking various control components, the SINUMERIK 810 GA3 was designed as a compact unit where the numerical control (NC), programmable logic controller (PLC), control keyboard and monitor are integrated into a single housing.





3.1.2 SINUMERIK 820 GA3





3.1.3 Overview of modules 810 GA3/820 GA3

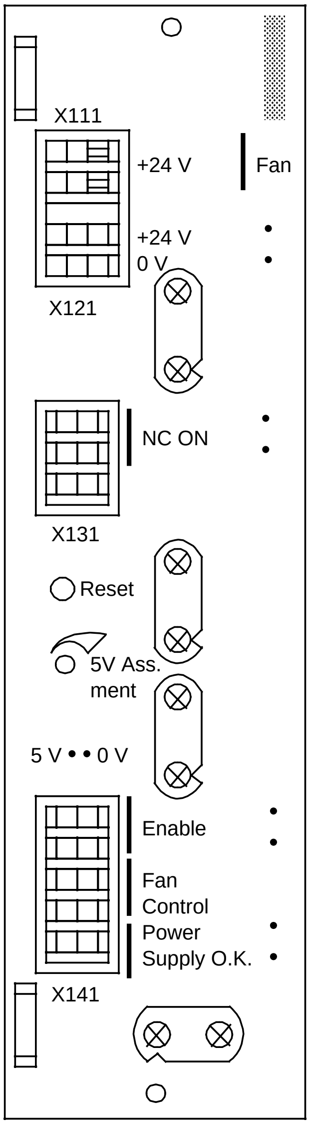
+ 24V DC power supply unit		6EV3 055-0BC
220V AC power supply unit (for SINUMERIK 820 GA3)		6EW1 861-3A..
CPU with ACOP		6FX1 138-5B...
Video interface module 7220 (monochrome and colour)		6FX1 151-1BA..
Video interface module (monochrome screen only)		6FX1 151-1BB
Interface module (orange terminal connector for sensor)		6FX1 121-2BC..
Memory module		6FX1 128-1BA..
EPROM submodule standard	- UMS 128 Kbyte	6FX1 128-4BB00
EPROM submodule	- UMS 128 Kbyte (N11)	6FX1 128-4BD00
EPROM submodule	- UMS 256 Kbyte (N12)	6FX1 128-4BC00
RAM submodule	- UMS 128 Kbyte (N13)	6FX1 126-6BA00
RAM submodule	- UMS 256 Kbyte (N14)	6FX1 135-3BA00
Measuring-circuit module (20mm wide) **DAC**		6FX1 121-4BA02
Measuring-circuit module (40mm wide - for EXE) **DAC**		6FX1 121-4BB02
HMS measuring circuit module		6FX1 145-6B..
Monitor adapter (brightness+contrast)		6FX1 120-0BB
I/O PCBs		6FX1 124-6AA01 6FX1 124-6AA02
MPG submodule for 3 handwheels		6FX1 126-5AA..
CUMPC interface		6FX1 132-1BA..
Input module	**for Mini EU**	6FX1 125-7BA..
Output module	**for Mini EU**	6FX1 122-8B..
CC interface for MPS	**for Maxi EU**	6FX1 132-1BB..
Power supply unit	**for Maxi EU**	6EW1 861-3A..
CU submodule interface (link module)		6FX1 136-2BA..
Master PLC link module (in the SINUMERIK)		6FX1 135-6BA00
S5 link module (in the SIMATIC)		6FX1 135-7BB00..

3.1.4 Power supply units

3.1.4.1 Power supply unit 24V DC (SINUMERIK 810 GA3)

6EV3 055-0BC

Brief description:



- Key for hardware reset on the front panel
- +5V adjustment potentiometer on the front panel
- +5V/0V measuring sockets (power supply unit output voltage) on the front panel
- Fan connection X111 (terminals on front panel)
- Fan OK signal (fan control) as floating relay output (100V/ 250mA; insulation voltage against housing 100V)
- Power supply unit OK signal as a floating relay contact (100V/250mA; insulation voltage against housing 100V) fed to front panel terminals.

When the control is delivered the fan control has to act directly on the power supply enable. This is achieved by jumpers of fan control with enable:

X141.1 with X 141.3 and X 141.2 with X 141.4

Both jumpers are inserted during manufacturing in the works.

Fan control can be used

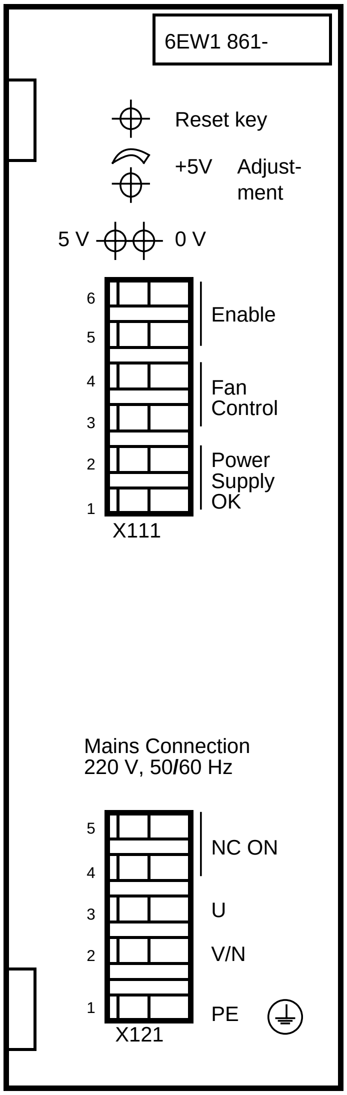
- As power supply enable

  Note: in this case the power supply is immediately shut down when the fan control responds.

- To trigger externally implemented shutdown logic for delayed shutdown of the power supply.

The power supply unit should be switched off by the fan control one minute at the latest after the monitoring response, otherwise the control may overheat and cause damage.

3.1.4.2 220V AC power supply unit (SINUMERIK 820 GA3)



Hardware reset key

+5V adjustment

Measuring sockets

6, 5 Terminals must be connected

4, 3 Fan OK signal

2, 1 Power supply OK signal

5, 4 NC ON

3, 2 220 V AC/50 Hz

Earth

Fan control can be used

- As power supply enable

  Note: in this case the power supply is immediately shut down when the fan control responds.

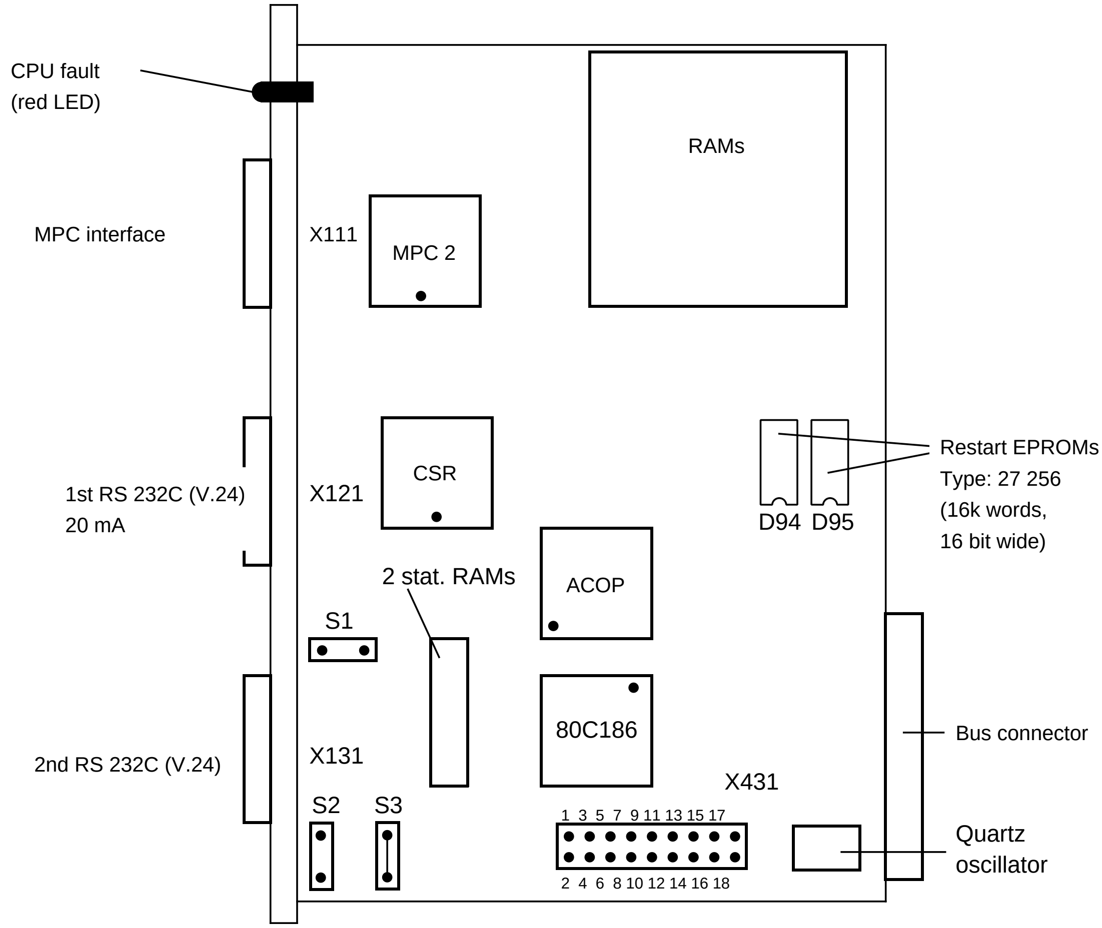
- To trigger externally implemented shut-down logic for delayed shutdown of the power supply.

The power supply unit should be switched off by the fan control one minute at the latest of the monitoring response, otherwise the control may overheat and cause damage.

3.1.5 CPU module

6FX1 138-5BB..

Clock frequency: 16 MHz



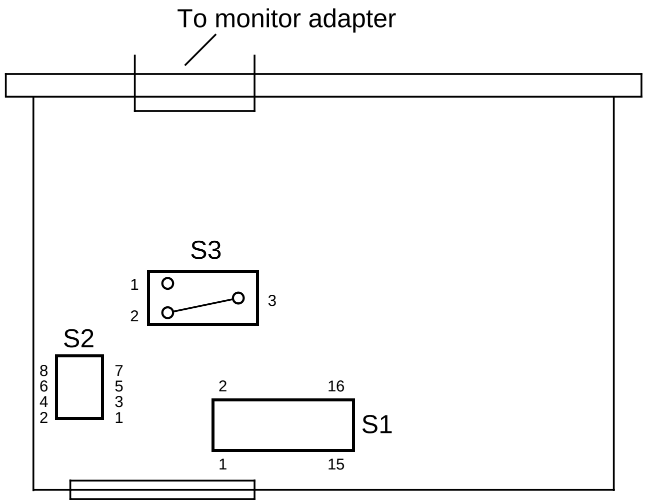
Quartz
oscillator

Standard jumpering: S1 open
S2 open
S3 closed

3.1.6 Video interface modules

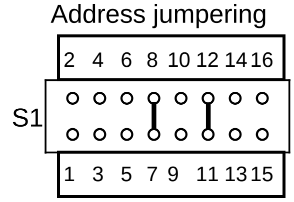
3.1.6.1 Video interface module, colour and monochrome

6FX1 151-1BA..
Can be used for colour and for monochrome monitor.



"DIP-FIX" switches S1, S2:

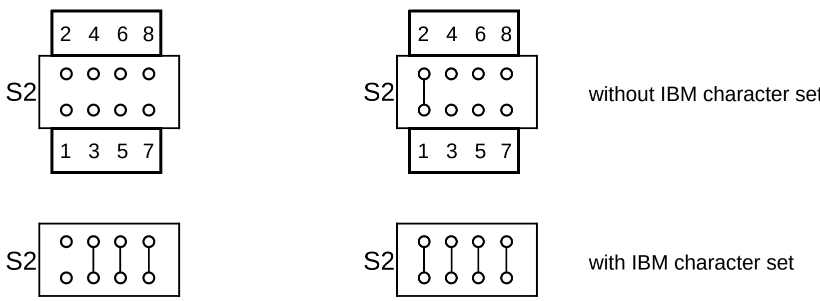
Address jumpering



Identification register

SINUMERIK 810/820
with monochrome screen

SINUMERIK 820
with colour screen



3.1.6.2 Video interface module, monochrome

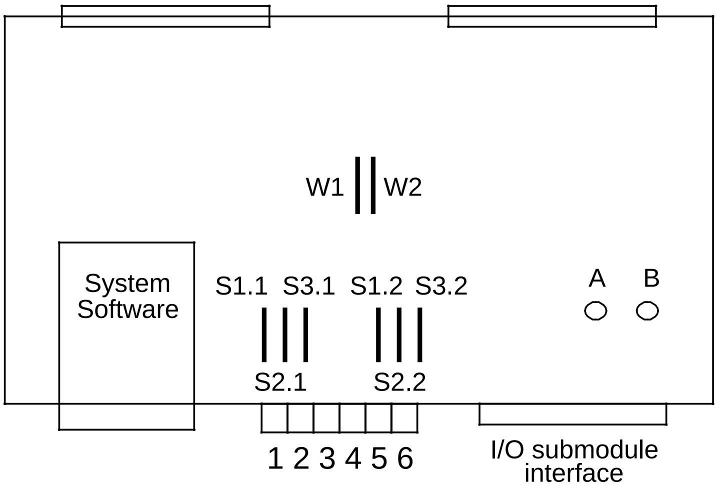
6FX1 151-1BB..

This module can only be used with a monochrome monitor.

3.1.7 (Vacant)

3.1.8 Interface module

6FX1 121-2BC..



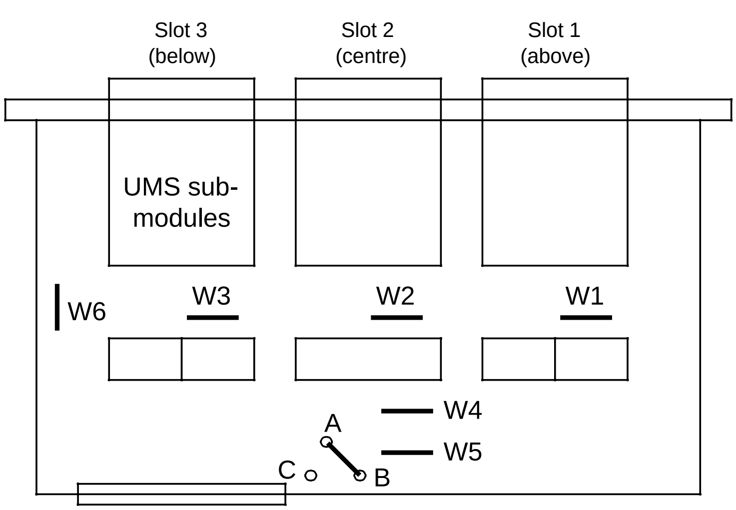
Module indentification:	W1	closed	Sensor 1:	S1.1 open
	W2	closed		S2.1 open
				S3.1 open
Soldering pins:	A—B	open	Sensor 1:	S1.2 open
				S2.2 open
				S3.2 open

See Section 7.3 for sensor coding.

3.1.9 Memory modules (only needed when UMS used)

A memory submodule may only be plugged in to slot 3.

- **Module for 3 x 256 Kbyte EPROM:** 6FX1 128-1BA..



Slots 1 and 2 must not be used.

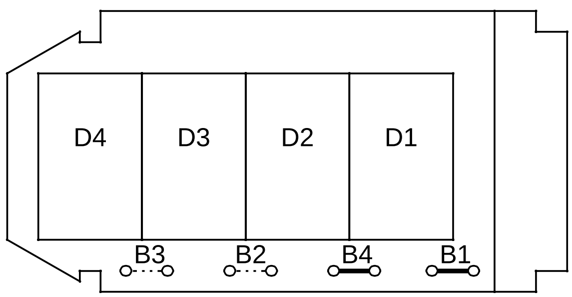
Jumpers:

W 1 closed – printed conductor
W 2 closed – module identification
W 3 closed – printed conductor
W 4 closed – module identification
W 5 closed – printed conductor
W 6 closed – printed conductor
A-B closed

If the UMS is pulled out when the control is switched off, it takes about 10 minutes for the UMS data to be deleted in the RAM. Only then does alarm 11 appear.

UMS modules for slot 3

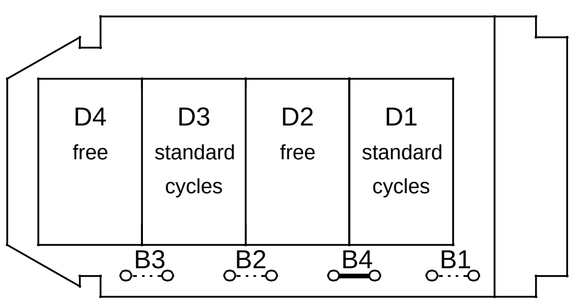
a) EPROM submodule: 6FX1 128-4BC00 (256 Kbytes)
6FX1 128-4BD00 (128 Kbytes)



Jumpers:

B 1 closed
B 2 open
B 3 open
B 4 closed

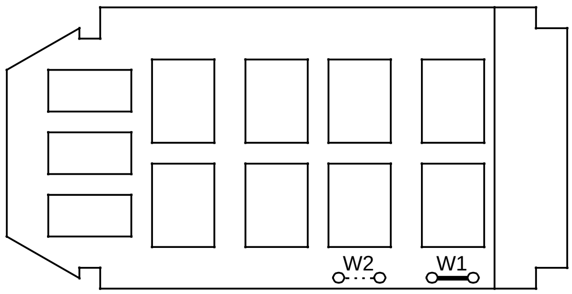
b) EPROM submodule: 6FX1 822-0BX03 (128 Kbytes, for T)
(Standard UMS, 6FX1 822-1BX03 (128 Kbytes, for M)
standard cycles)



Jumpers:

B 1 open
B 2 open
B 3 open
B 4 closed

c) RAM submodule: 6FX1 126-6BA00 (128 Kbytes)

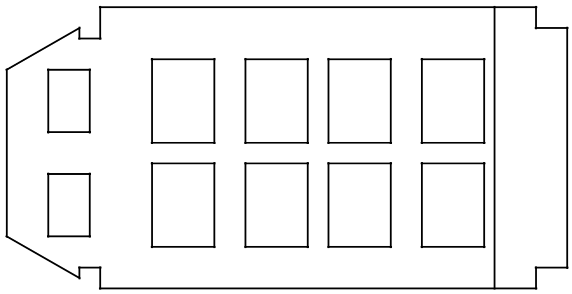


Jumpers:

W 1 closed
W 2 open

Components on front and rear.

d) RAM submodule: 6FX1 135-6BA00 (256 Kbytes)



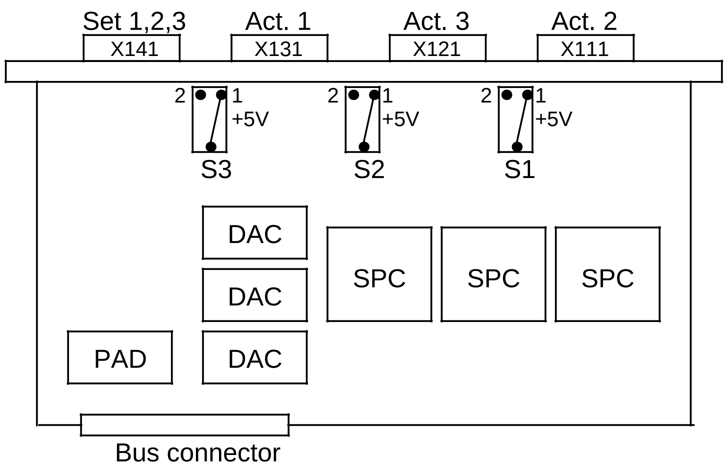
Note:

No jumpers to be taken into account!

3.1.10 Measuring-circuit modules

3.1.10.1 SPC measuring circuit module 6FX1 121-4BA . . (standard measuring circuit module)

For analog setpoint output DAC: 6FX1 121-4BA . . 20 mm
6FX1 121-4BB . . 40 mm (EXE)

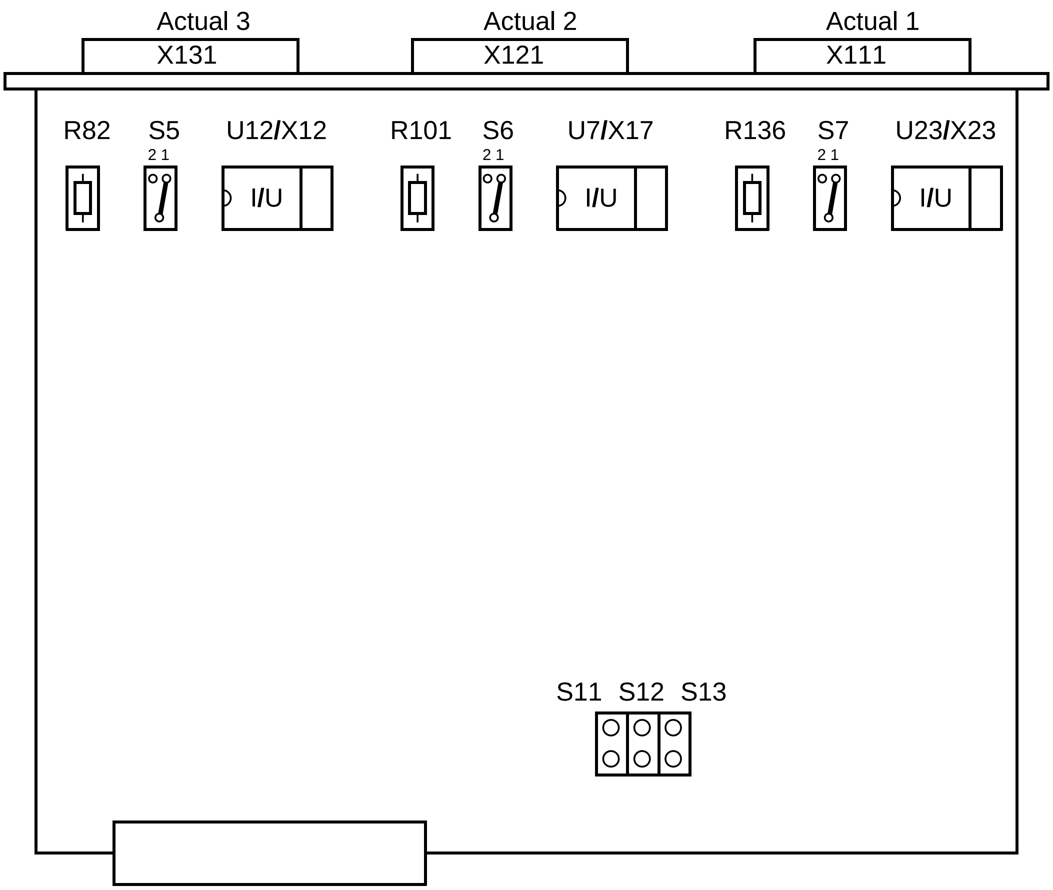


Interface description:

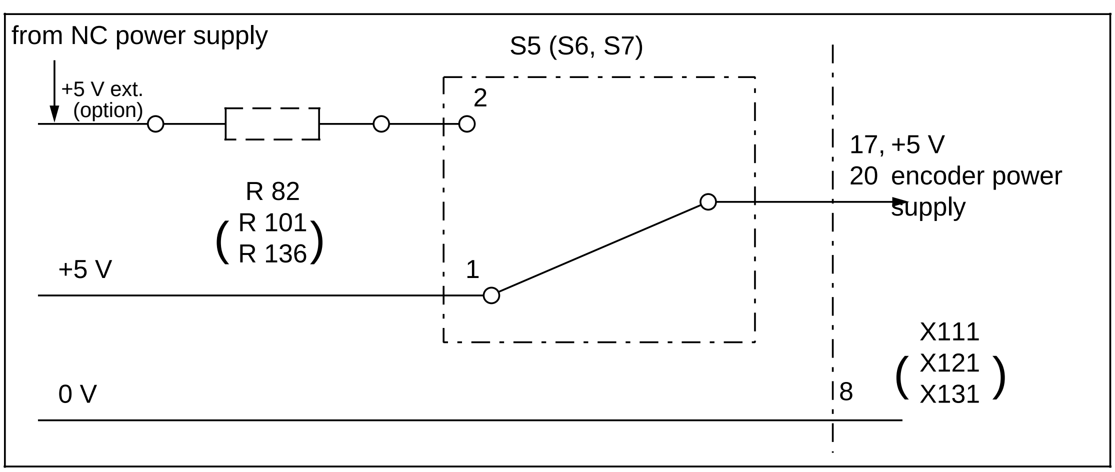
- 3 actual values
- 3 analog setpoints (command values)
- No NC-RDY relays
- No servo enable input
- Non-floating servo enable

3.1.10.2 HMS measuring circuit module 6FX1 145-6B 00

=A	with submodule interface	40 mm wide
=B	without submodule interface	20 mm wide



Encoder power supply:

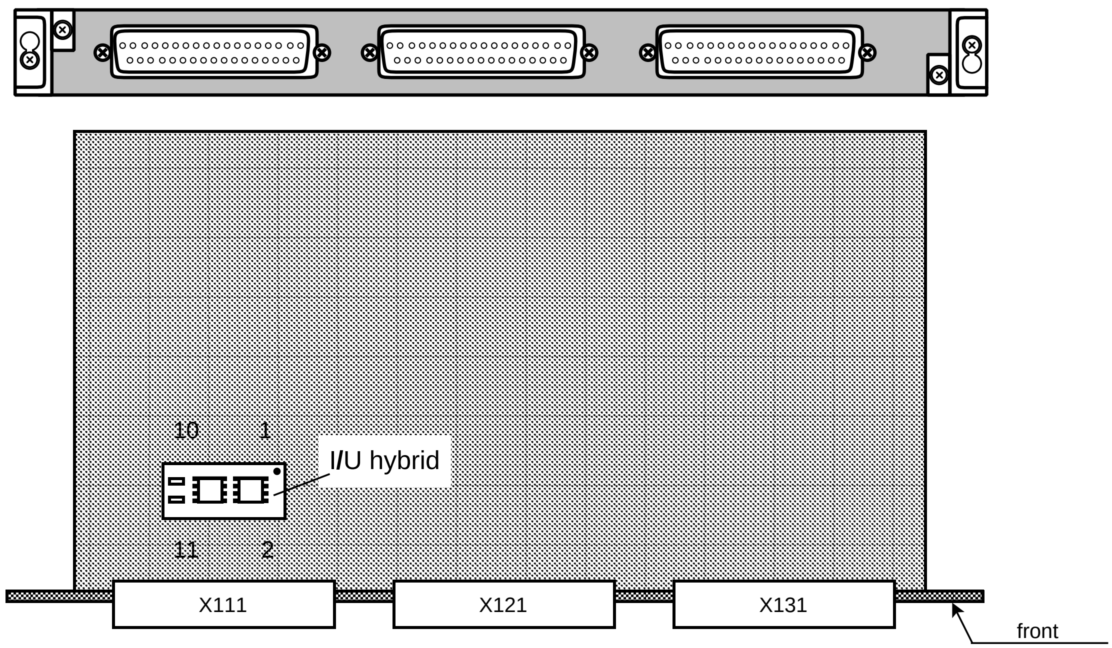


Jumpers S11, S12 and S13 are open.

Each of the two HMS measuring circuit modules contains 3 measuring circuit/actual value inputs for connecting 3 axes with incremental position encoders. On the double-width module, 3 additional submodule slots are provided into each of which 1 analog measuring circuit setpoint submodule (HMS setpoint submodule) or 1 SIPOS absolute encoder submodule+1 measuring circuit setpoint submodule can be plugged. Both modules can be combined such that the double-width HRS measuring circuit module contains the measuring circuit setpoint submodules for all 6 axes/spindles (in this case, however, no SIPOS absolute encoder submodule can be inserted).

Note:

- The SIPOS absolute encoder submodule can be inserted only in the upper 2 submodule slots of the HMS measuring circuit module.
- An I/U hybrid that can be plugged on the module enables also linear scales with unconditioned current signals to be connected to the HMS module. Each of the 3 measuring system inputs on the HMS module can be provided with an I/U hybrid.



The I/U hybrid is inserted into the socket with pins 2 and 11 pointing to the front as shown above. The colour dot on the hybrid marks pin 1.

I/U hybrid (I/U: current/voltage conversion)

The HMS measuring circuit module can be used only for SIPOS encoders with unconditioned voltage signals as standard. Linear scales with unconditioned current signals can also be used as options if a I/U hybrid is inserted into the relevant jumpering socket (U12, U17, U23) in place of the short-circuiting connector. This direct connection is only possible up to a lead length of 18m for reasons of interference immunity. For lead lengths exceeding 18 m a converter box converting the current signals into voltage signals must be provided. The I/U hybrid must be installed as described in the Installation Instructions.

Order number:	
I/U hybrid:	6FC3 988-7CN

Amplifier for unconditioned current signals (l>18 m)	
Order number:	6FC9 320-4HM12

Encoder signal routing interface

Connector designations U12 (X12), U17 (X17), U23 (X23)

20-pin IC socket (DIL 20) used to connect a current/voltage converter hybrid when employing encoders with unconditioned current signals (option).

Order number: 6FC3 988-7CN

+5V	11	. . . . .	10	+5V
-15V	12	. . . . . .	9	-15V
0V	13	. . . . . .	8	0V
+15V	14	. . . . . .	7	+15V
RN	15		6	R1N
RP	16		5	R1P
BN	17		4	B1N
BP	18		3	B1P
AN	19		2	A1N
AP	20		1	A1P

A1P	Encoder signal	A	(Unconditioned voltage or current signal)
A1N	Encoder signal	*A	(Unconditioned voltage or current signal)
B1P	Encoder signal	B	(Unconditioned voltage or current signal)
B1N	Encoder signal	*B	(Unconditioned voltage or current signal)
R1P	Encoder signal	R	(Unconditioned voltage or current signal)
R1N	Encoder signal	*R	(Unconditioned voltage or current signal)

AP	Encoder signal	A	(Unconditioned voltage signal)
AN	Encoder signal	*A	(Unconditioned voltage signal)
BP	Encoder signal	B	(Unconditioned voltage signal)
BN	Encoder signal	*B	(Unconditioned voltage signal)
RP	Encoder signal	R	(Unconditioned voltage signal)
RN	Encoder signal	*R	(Unconditioned voltage signal)

When using encoders supplying unconditioned voltage signals (standards) (SIPOS encoders), the interfaces are fitted with short-circuiting encoding connectors (X12, X17, X23) connecting opposite pins (1-20, 2-19, 3-18, ...). The power supply pins are already interconnected on the module.

HMS setpoint submodule 6FX1 132-5BA00

The submodule is plugged on the relevant interface in the HMS measuring circuit module 6FX1 145-6BA00 (double width with submodule interface). Setpoints of up to 3 axes/spindles can be output via a submodule. Up to 2 HMS setpoint submodules can be plugged in the HMS measuring circuit module. The additional submodule can be used only in conjunction with the HMS measuring circuit module 6FX1 145-6BB00 (single width without submodule interface). For the SINUMERIK 810 this results in the following maximum configuration (with SINUMERIK 820 more space is available in the subrack):

1 HMS measuring circuit module with submodule interface
1 HMS measuring circuit module without submodule interface
2 HMS setpoint submodule

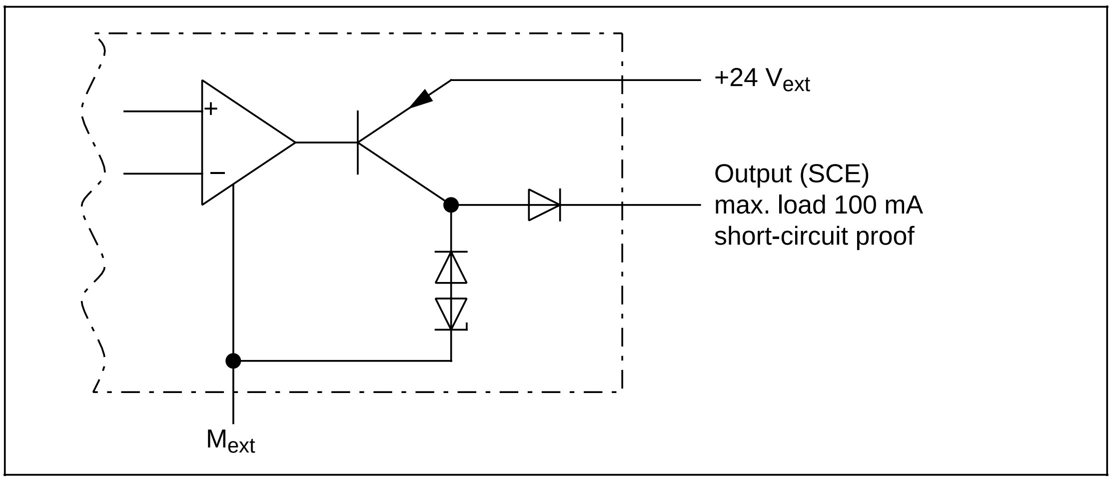
and the following minimum configuration when using HMS:

1 HMS measuring circuit module with submodule interface
1 HMS setpoint submodule

Limitation:

If a SIPOS absolute encoder submodule is plugged into the HMS measuring circuit module, only 1 HMS setpoint submodule can be additionally inserted into this module.

Output for servo enable



SIPOS absolute encoder submodule G33961-A3729-L1

1 SIPOS absolute encoder submodule can be inserted into the double-width HMS measuring circuit module using the upper 2 submodule slots. This allows simulation of absolute measuring values for all 3 inputs. In this case, however, only 1 HMS setpoint submodule can be plugged at the same time. A battery submodule can be plugged on the SIPOS absolute encoder submodule to store absolute values in the event of power failure.

SIPOS absolute encoder 6FC9 320-3CT

Multiturn absolute encoder with 2500 increments.

This includes the cable 6FC9 344-4D .

The SIPOS encoder system is a multiturn absolute encoder working on an absolute basis at POWER ON and on an incremental basis during operation.

To reduce the size of the encoder, part of its electronics has been arranged on another submodule, i. e. the absolute submodule.

The control requests from the absolute encoder the absolute value of the encoder after every POWER ON and then calculates the absolute machine position.The absolute encoder then acts as a standard incremental encoder with amplified sine/cosine output signals.

The absolute information is derived from the encodered dial information as a cyclically absolute value and the reading of the battery-backed revolutions counter. The backup battery covering 3 axes is located on the absolute submodule.

Basic encoder data:

Max. sampling frequency:	500 kHz
Dial increments:	2500
Revolutional information:	16 bits
Starting accuracy:	1 µm, with 10 mm spindle pitch (10 000 increments/rev)
Error information:	8 bits
Backup period:	5800 hours for 3 encoders, own battery monitor

The condition of the backup battery is checked every 10 minutes also during operation and an error message is displayed in the control in the event of malfunction.

An additional backup in the encoder enables the measuring circuit module or the encoder cable to be exchanged without absolute information being lost.

This backup can be maintained over a period of 5 hours.

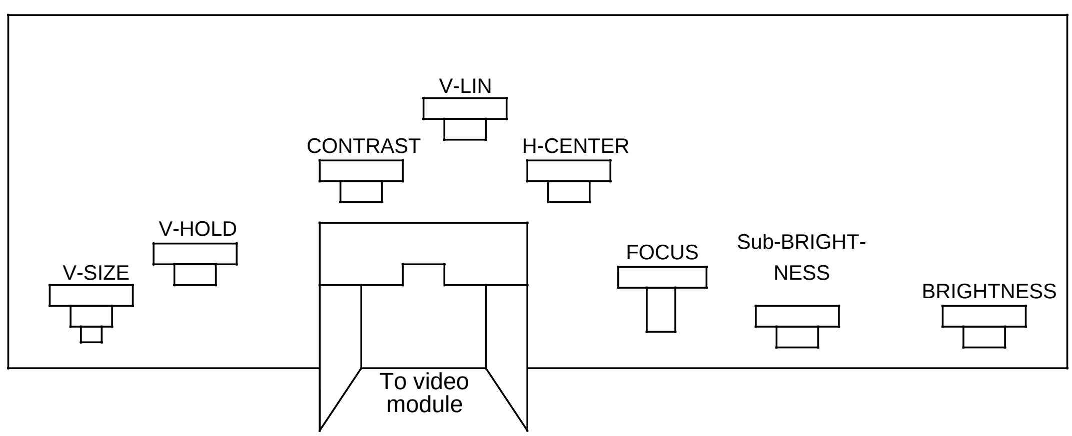
SIPOS unconditioned signal encoder 6FC9 320-CS

Multiturn incremental encoder with 2500 pulses/rev, limiting frequency 500 kHz. The cable 6FC9 344-4D is included.

3.1.11 Monitor control unit

This module is part of the **monitor**.

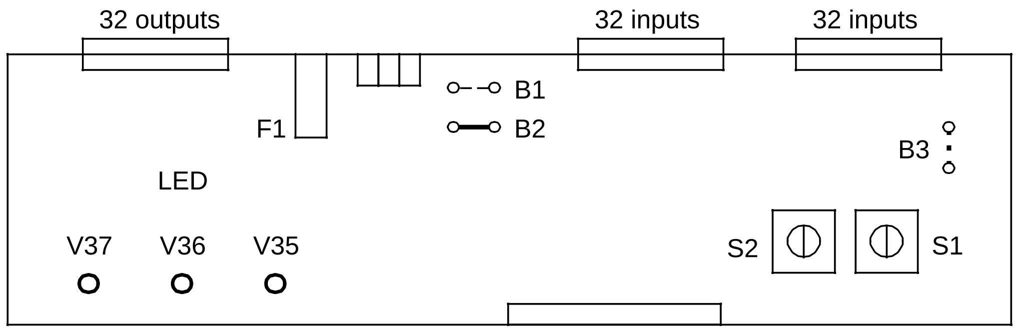
The monitor has already been adjusted in the works. If the brightness must be adapted to a special working environment, this can be done with the potentiometer ”BRIGHTNESS”.



3.1.12 I/O submodules

Up to 4 submodules can be connected to the interface module.
Submodule with 2 input connectors: 6FX 1124-6AA01

- 64 (2 x 32) inputs 24V
- 32 outputs :
  – 24 outputs each 400 mA, 24V DC, short-circuit-proof (bytes 0 - 2)
  One of the red LEDs V35, V36 or V37 flashes when short-circuit is indicated.
  – 8 outputs each 100 mA, 24V DC, not short-circuit-proof (byte 3)

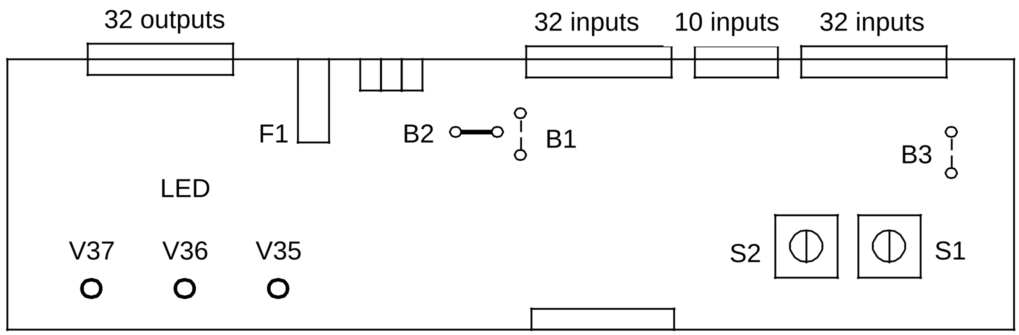


Jumpers:

B 1 open F1 6.3 A fuse (for 24V)
B 2 closed
B 3 open

Submodule with 3 input connectors*:* 6FX 1124-6AA02 (Retrofit order no.: 6FC3 984-3RB
(Basic submodule with mounting plate, Retrofit order no.: 6FC3 984-3RA)

- 32 inputs 24V
- 32 inputs 24V (including 10 double-function inputs machine control panel for 2 axis selector switches)
- 32 outputs:
  24 outputs each 400 mA, 24V, short-circuit-proof (bytes 0-2)
  One of the red LEDs V35, V36 or V37 flashes when short-circuiting is indicated.
  8 outputs each 100 mA, 24V, not short-circuit-proof (byte 3)
- Voltage applied to+24V and M-out.

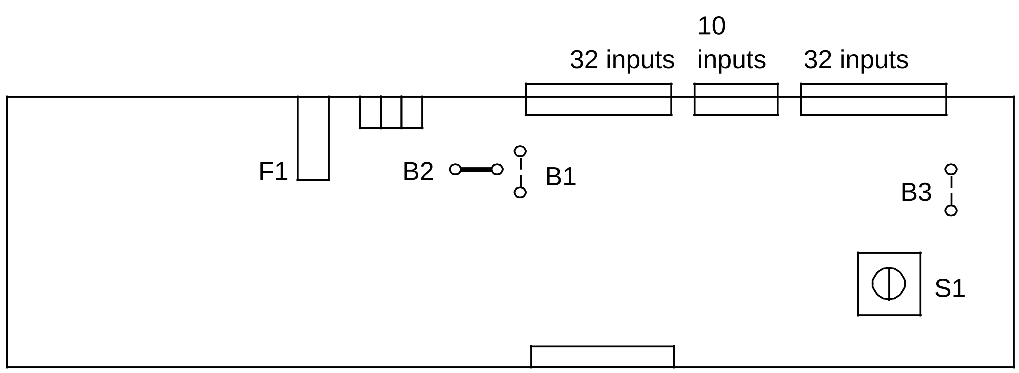


Jumpers:

B1 open
B2 closed
B3 open
F1 6.3 A fuse for outputs and external measuring systems

Submodule with 3 input connectors: 6FX1 124-6AB02 (Retrofit order no.: 6FC3 984-3RC)
(Basic submodule with mounting plate, Retrofit order no.: 6FC3 984-3RD)

- 32 inputs 24 V
- 32 inputs 24 V (including 10 double-function inputs machine control panel for 2 axis selector switches)
- Voltage applied to+24V and M-out.

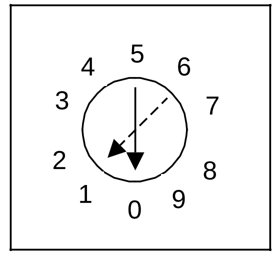


Jumpers:

B1 open
B2 closed
B3 open
F1 6.3 A fuse for external measuring systems

Switch position for S1 / S2 on individual submodules

	Sub-module 1	Sub-module 2	Sub-module 3	Sub-module 4
S1	0	1	2	3
S2	0	1	2	3



S1 / S2

S1: Address setting for inputs
S2: Address setting for outputs

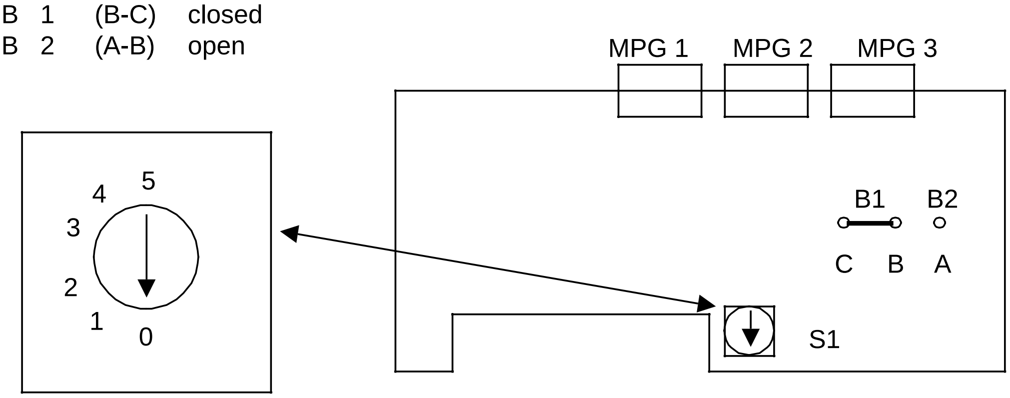
Determination of location of short-circuit

Red LED	V37	V36	V35	No LED
Submodule 1	QB 0	QB1	QB 2	QB 3
Submodule 2	QB 4	QB 5	QB 6	QB 7
Submodule 3	QB 8	QB 5	QB 10	QB 11
Submodule 4	QB 12	QB13	QB 14	QB 15
Max. current	400 mA	400 mA	400 mA	100 mA

3.1.13 Handwheel submodule (MPG)

6FX1 126-5AA . .

B 1 (B-C) closed
B 2 (A-B) open



S1 always in position 0

Jumper B2 must not be closed, otherwise handwheel operation is not guaranteed.

3.2 P06 expansion unit (Mini EU)

3.2.1 Overview of modules

Designation	Order No.	Order code	1	2	3	4	5	6	7	8	9	10	11	12
BUS / MINI EU	570348 9001 00													
CU/MPC interface with power supply	6FX 1132-1BA01	–												
I-interf., 32 inputs 20 mm	6FC9 320-8AA 6ES5 432-3BA11	N60												
O-interf., 32 outp. 0.5 A 20 mm	6FC9 320-8AB 6ES5 445-3AA12	N65												
O-interf., 16 outp. 2 A 40 mm	6FC9 320-8AC 6ES5 444-3AA12	N70												
I-interf., 32 inputs 20 mm	6FC3 988-4DB 6ES5 420-4UA11	N61												
O-interf., 32 outp. 0.5 A 20 mm	6FC3 988-4DG 6ES5 441-4UA11	N66												
O-interf., 16 outp. 2 A 40 mm	6FC3 988-4DJ 6ES5 454-4UA11	N68												
I-interf., 16 inputs 20 mm	6FC3 988-4DK 6ES5 432-4UA11	N69												
I-interf., 64 inputs 20 mm	6FC3 986-4DM 6FX1 125-7BA00	N71												
O-interf., 32 outp. 0.5 A 20 mm	6FC3 986-4DN 6FX1 122-8BA01/ 8BA01	N72												
O-interf., 32 outp. 2 A 40 mm	6FC3 986-4DP 6FX1 122-8BB01/ 8BB01	N73												

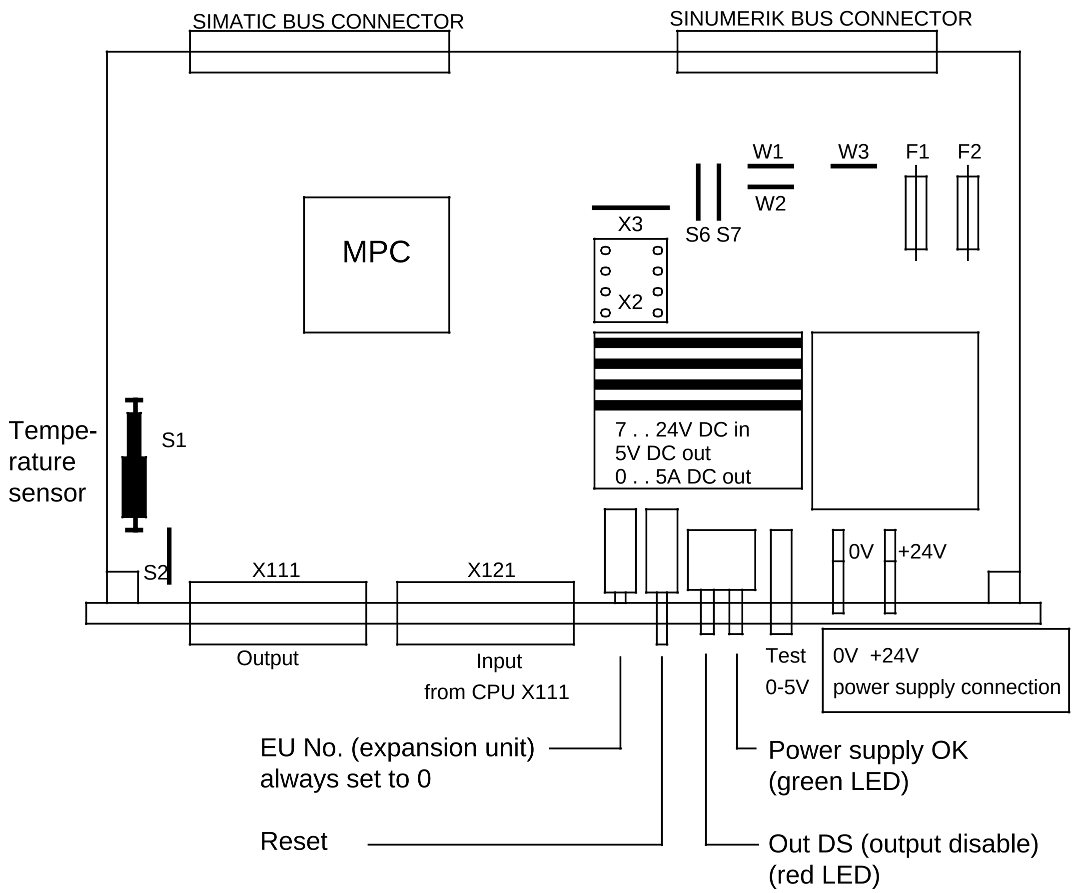
Basic version Option

A description of modules N61, N66, N68 and N69 can be found in the corresponding SIMATIC Product Manual.

3.2.2 Jumperings

3.2.2.1 Interface CU/MPC

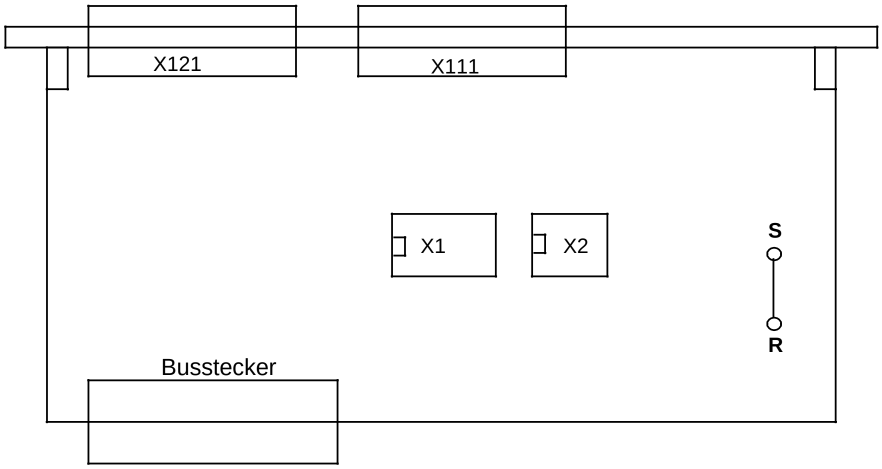
Interface module for mini EU : 6FX 1132-1BA . .
(- 1BB . . without power supply for maxi EU)
always at slot 1 of mini EU



F1 4.0 A power supply
F2 1.6 A fan fuse
W1 closed: (test panel)
W2 - internal printed conductor
W3 - internal printed conductor
X3 closed: OUT-DS from central unit (CPU) active
S1 - temperature switch
S2 closed: shutdown in the event of over-temperature after 20 sec.
S6 closed: (test panel)
S7 closed: (test panel)

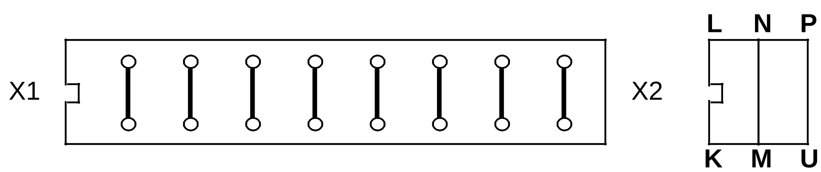
3.2.2.2 Input module for Mini EU:

6FX1 125-7BA.. 64 inputs



Jumper: S-R closed

Address setting on the DIP-FIX **socket** X1 and X2



Input byte	8	16	32	64	128	W1	W2	W3
0-7								
8-15	x							
16-23		x						
24-31	x	x						
32-39			x					
40-47	x		x					
48-55		x	x					
56-63	x	x	x					

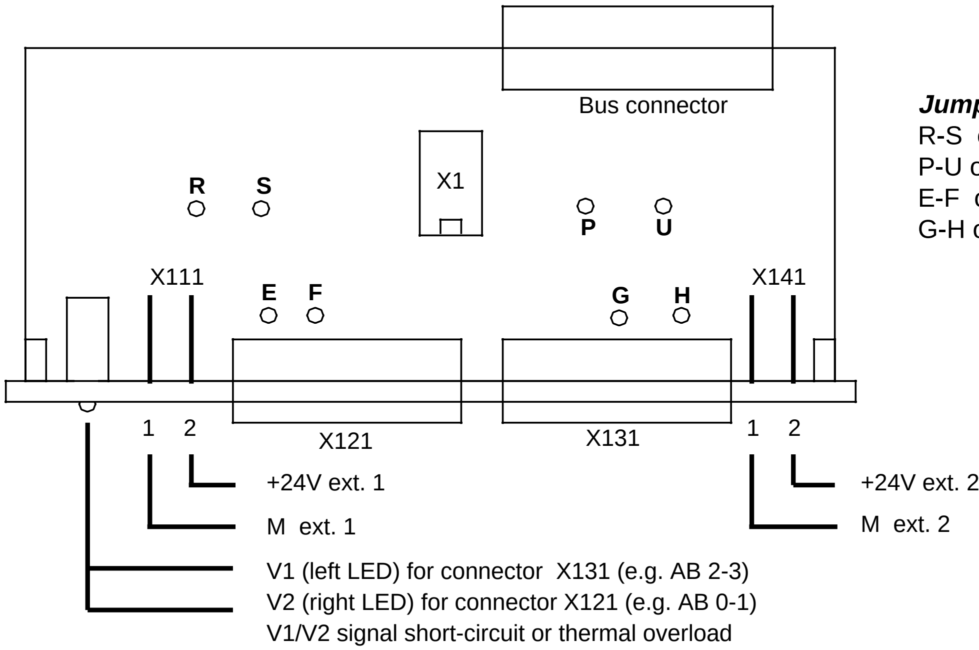
x: DIP-FIX closed

All DIP-FIX of X2 are always closed

Connector X111:
e.g. IB0-3
Connector X121:
e.g. IB4-7

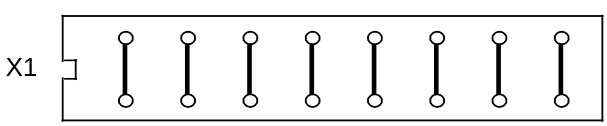
3.2.2.3 Output module for Mini EU:

6FX1 122-8B.. 32 outputs 0.5 A/2.0 A



Jumpers:
R-S open
P-U open
E-F open
G-H open

Address setting on the DIP-FIX socket 1



Output byte	4	8	16	32	64	128	A	C
0 ... 3								
4 ... 7	x							
8 ... 11		x						
12 ... 15	x	x						
16 ... 19			x					
20 ... 23	x		x					
24 ... 27		x	x					
28 ... 31	x	x	x					

x: DIP-FIX closed

The outputs are short-circuit-proof against earth and are switched off separately.

3.3 P08 expansion unit (Maxi EU)

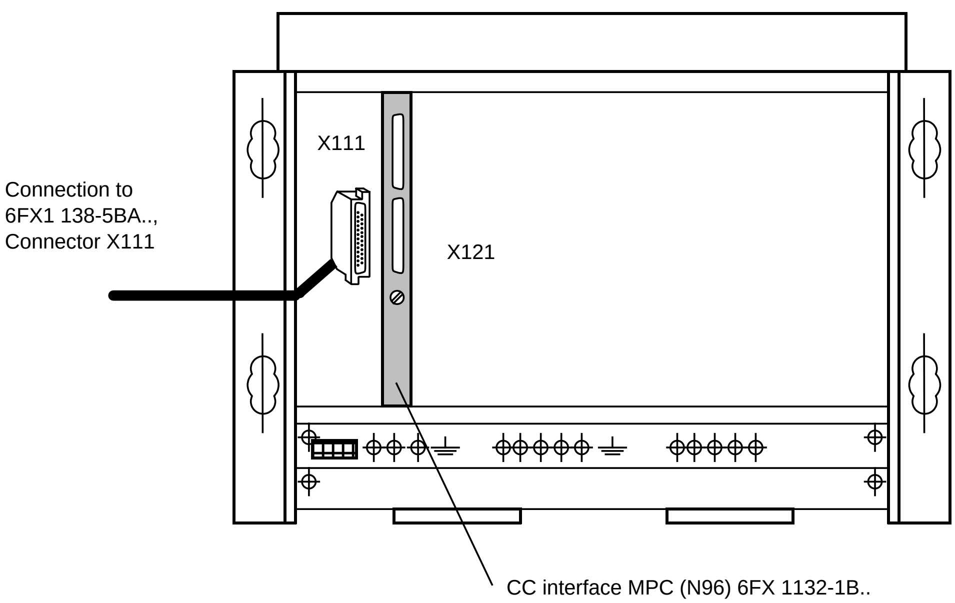
3.3.1 Overview of modules

Designation	Order No.	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24
SIMATIC bus connectors																									
SINUMERIK bus connectors																									
Power supply unit	6EW1 861-3A																								
CC interface MPC	6FX 1132-1BB01																								
Digital input module 64 inputs N71	6FX1 125-7BA00																								
Digital output module 0.5A 32 outputs N72	6FX1 122-8BC01																								
Digital output module 2A 32 outputs N73	6FX1 122-8BD01																								
SIMATIC I/O Single width (20mm) N61, N66																									
SIMATIC I/O Double width (40mm) N68, N70																									

Basic version   Option   Alternative

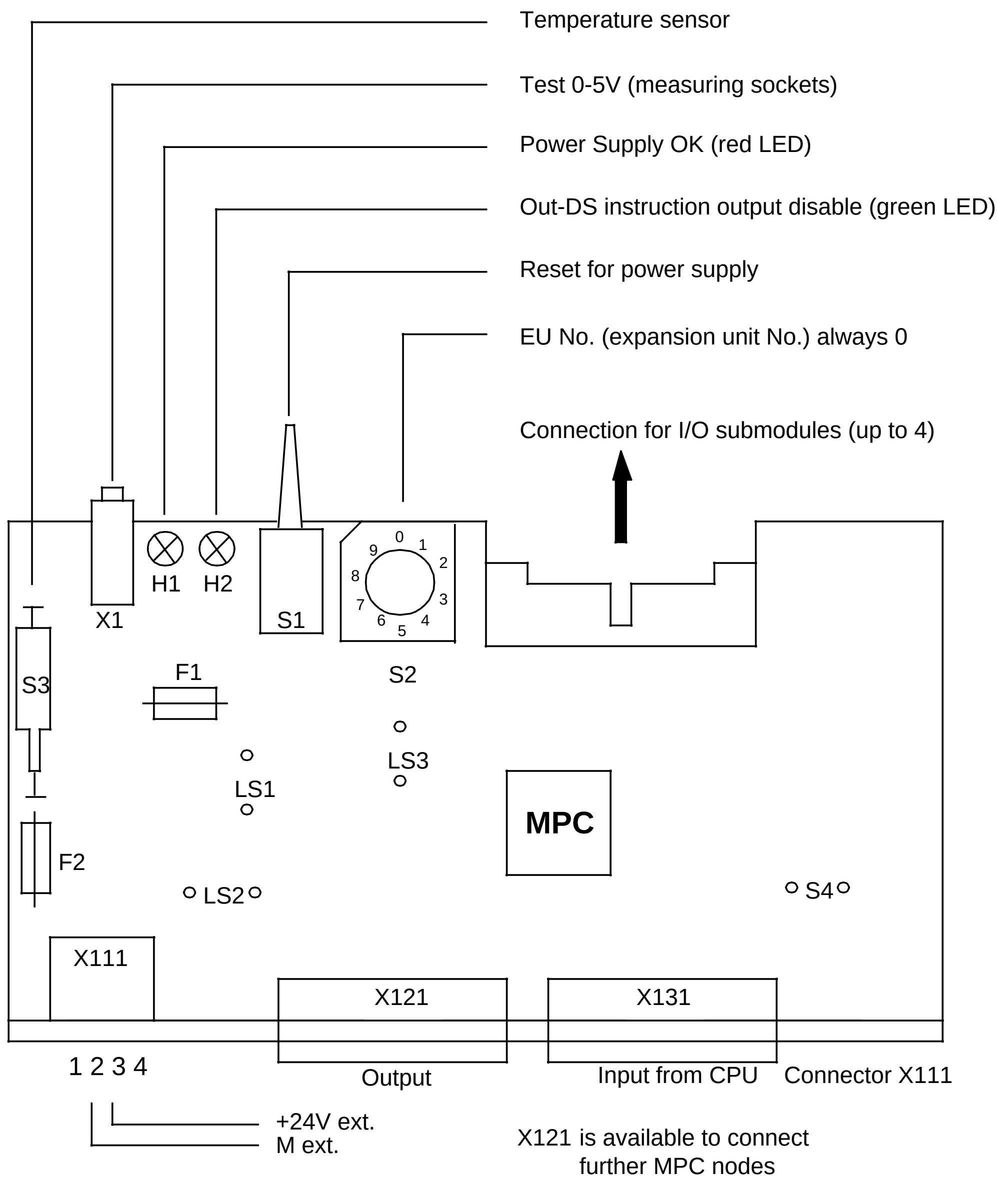
A description of the SIMATIC I/O devices can be found in the corresponding SIMATIC Product Manual.

3.3.2 PLC expansion unit (P08)



3.4 Interface CU modules

Distributed connection for I/O submodule 6FX 1136-2BA..
(Handwheel submodule can **not** be connected)

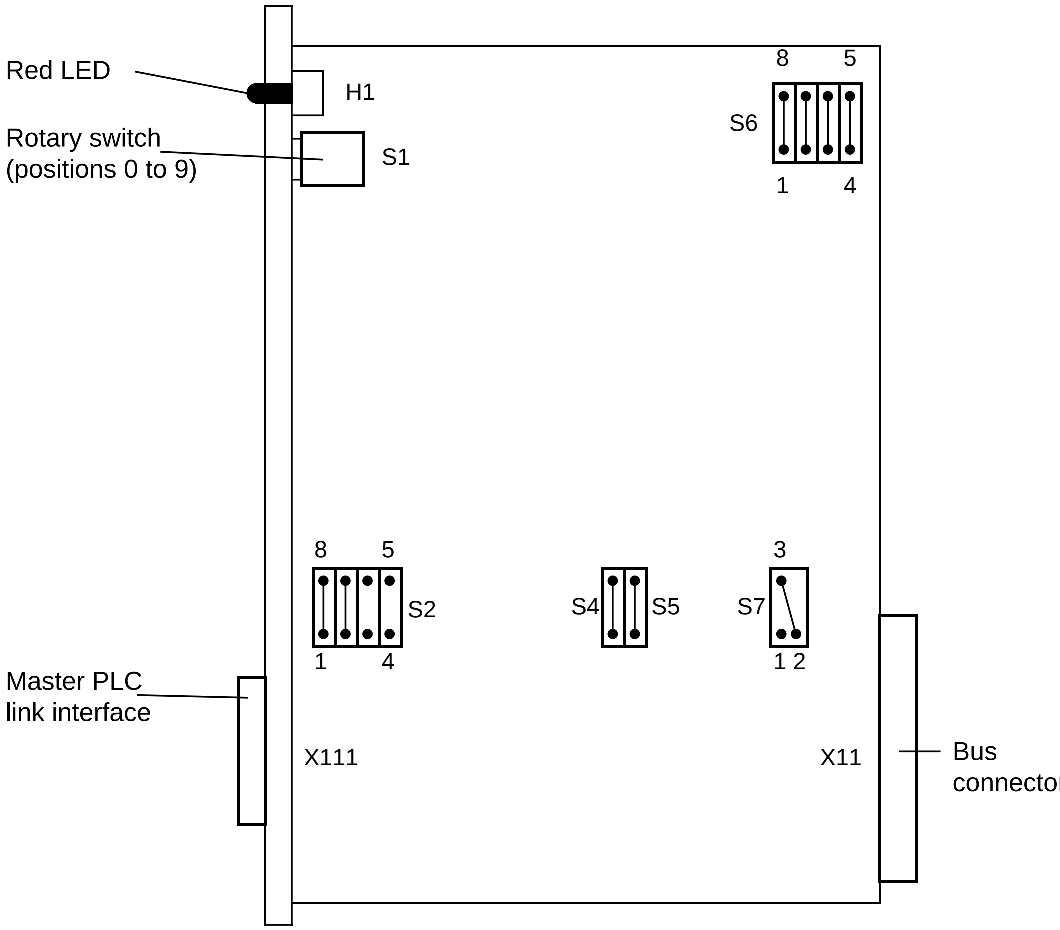


F1 fuse 2.0A slow
F2 fuse 1.6A fast (power supply)
S4 closed (for test panel)
LS1 open (closed: connect M ext. to shield)
LS2 open (closed: input +24V via X111 Pin 1)
LS3 closed (for test panel)

3.5 Master PLC link modules

3.5.1 Master PLC link module (in the SINUMERIK)

6FX1 135-6BA

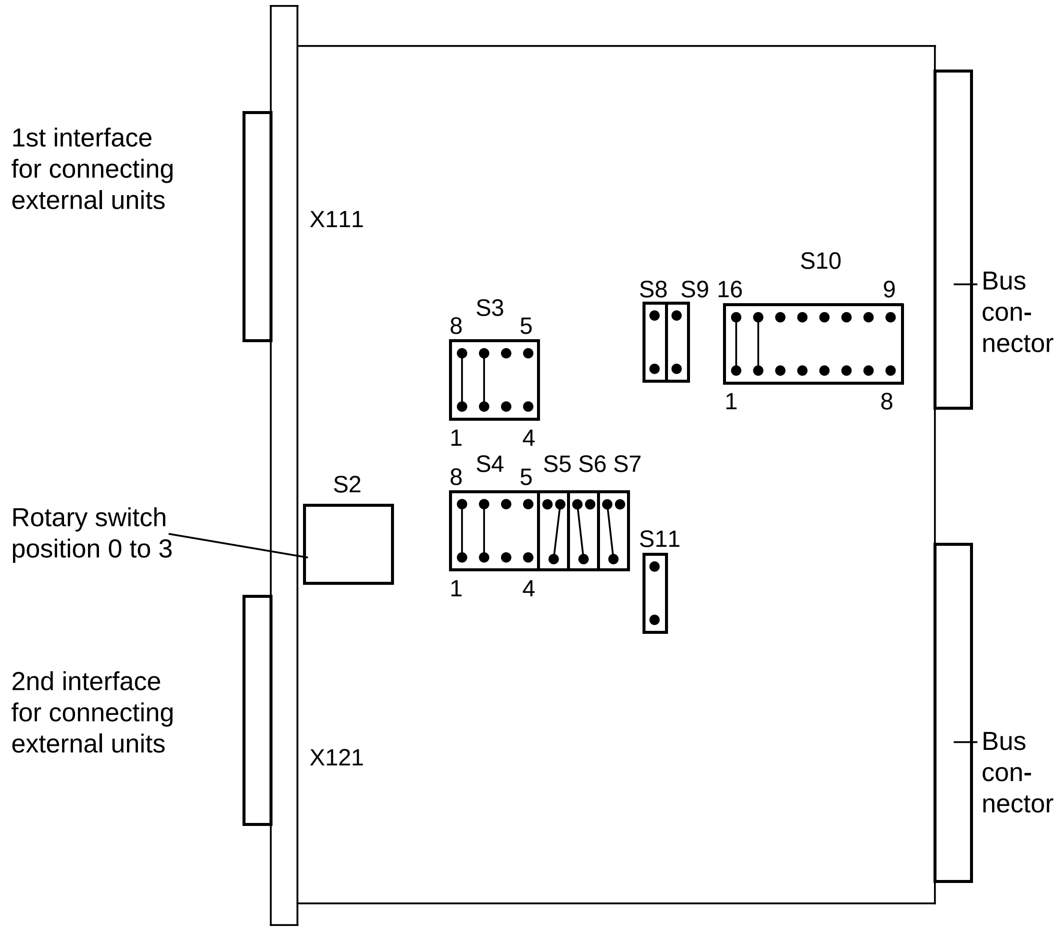


Standard jumpering:

S1	position 1	
S2	1—8 closed	3—6 open
	2—7 closed	4—5 open
S4	closed	
S5	closed	
S6	1—8 open	
	2—7 open	
	3—6 open	
	4—5 open	
S7	2—3 closed	

3.5.2 Master PLC link module (in the SIMATIC)

6FX1 135-7BB...



For jumpers see the description of the master PLC link in Section 10.11

4 Voltage and Functional Tests/Drift

4.1 Voltage test

4.1.1 Voltage supply

The connection conditions and voltage supply limit data should be checked **prior to** powering up.

SINUMERIK 810 GA3

Power supply unit +24 V DC

- Voltage 20 V ... 30 V incl. ripple
- Power 120 W

SINUMERIK 820 GA3

Power supply unit 230 V AC

- Voltage 187 V ... 235 V
- Frequency 48 Hz ... 63 Hz
- Power 500 VA

4.1.2 Limit temperature

In addition, the fan is monitored by means of an NTC thermistor (at the power supply unit).

4.1.3 Direct voltage +5 V

The voltage is already set at the factory to approx. +5.15 V (on account of the line voltage drops).

4.2 Functional test

4.2.1 CPU monitor

The LED on the CPU (6FX 1138-5BA01) provides information as to the status of the control.

When the monitor responds (LED lights up), the axes are shut down and the PLC outputs are disconnected.

During run-up of the control (Power On routines) the LED flashes repeatedly.

LED continues to indicate after Power On routines:

- CPU error
- EPROM error
- Incorrect machine data
- Incorrect module jumpering
- Defective module or rear panel

LED lights up during operation:

- Hardware fault on a module
- CPU is in a loop
- CPU previously in a loop for a lengthy period, causing the monitor to respond.

4.2.2 EPROM CHECK

A cross-check sum test of the system program memory (EPROM) is performed whenever the control is switched on (prior to the POWER ON routines). In the cyclic mode, a parity check is performed.

If a difference is found between the set sum and the actual sum, the control displays the text ”**CHECKSUM ERROR SYSTEM MODULE**” on the screen. If a difference is found in the cyclic mode, the text ”**TIME OUT IF PARITY**”is displayed together with the contents of the CPU register.

If the system module is not inserted, the following message appears on the screen:

WRONG EPROM SYSTEM MODULE

The system identifier can be read out in overall reset mode (overall reset display)

Meaning:

3GE 570.822.9602.30

Release 0

Software version 3

02 Language:

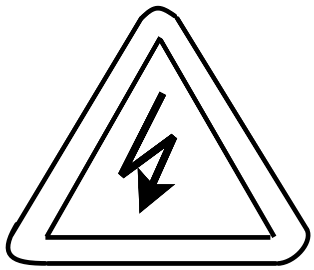
00	English/German
02	French/German
03	Italian/German
04	Spanish/German
05	Dutch/German
06	Russian/German
07	Swedish/German
08	Finnish/German
10	Polish/German
11	Hungarian/German
12	Czech/German
13	Turkish/German
20	French/English
21	Italian/English
22	Spanish/English

96 Type of control:

96	T Version
97	M Version

805 . . . . . System 810 GA1 and not configurable
818 . . . . . System 810 GA1 and configurable (option N30)
821 . . . . . System 810 GA2
822 . . . . . System 810 GA3/820 GA3
570 . . . . . SINUMERIK

4.2.3 Brightness adjustment on screen



C A U T I O N !

High voltage approx. 16 kV in visual display unit and at high-voltage transformer, anode lead and anode terminal at cathode ray tube.

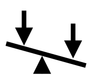
The brightness can be adjusted using a potentiometer located in the CRT unit and accessible from the outside.

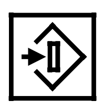
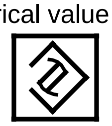
The contrast may also be adjusted at the monitor adapter using a 10-turn potentiometer (see Section 3). However, this potentiometer is ***not*** accessible from the outside.

Correct adjustment of the contrast is usually performed at the factory.

4.3 Drift compensation

Drift compensation can be performed when the axes (of the CNC) and the drives are in closed-loop control.

If the "  " fails to go out after a traversing movement, the drift has exceeded the maximum permissible value and drift compensation **must** be performed.

Operator input:	-	Softkey "DIAGNOSIS"		
	-	key >		
	-	Softkey: "NC-COMM"		
	-	Numerical value 2720+key		1st axis
	or	numerical value 2721+key		2nd axis
	or	numerical value 2722+key		3rd axis
	or	numerical value 2723+key		4th axis
	or	numerical value 2724+key		5th axis
	or	numerical value 2725+key		6th axis
	or	numerical value 2726+key		7th axis
	-	EDIT  key		
	.			
	.			
	.			
	Possibly two or more axes			
	.			
	-	RECALL ∧ key		

Drift compensation must be performed individually for all axes.

In the case of very high-precision machines drift compensation must be performed several times daily. This is because of the variations in temperature during operation and because drift is a direct factor in terms of the following error.

5 Standard Installation

5.1 Standard installation sequence

In the following installation sequence it is assumed that neither the NC nor PLC section of the control has already been started up.

Section 1 (Preconditions and visual inspection) of the Installation Guide must be observed.

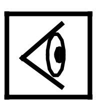
All cables and modules, in particular all I/O modules (6FX 1124-6AA . .), must be inserted or connected.

The NC MD and MD bits of importance for the PLC will be dealt with in Section 6.3.2.

On first installation it is important to check that the control was supplied from the factory with the option bits set as ordered.
If the machine data were deleted or the standard machine data loaded the option bits are also deleted.

The installation sequence below is mandatory since the NC and PLC can no longer be clearly differentiated in the case of the SINUMERIK 810 GA3/820 GA3. A ”V” sign is generated as a positive response to functions initiated by means of a softkey.

Notes on first installation:

1. Press  key and switch on control simultaneously. Keep the key depressed until the OVERALL RESET display appears on the screen.

2. Press ”MACH. DATA” softkey

2.1 Press ”CLEAR NC MD” softkey
NC MD are cleared

2.2 Press ”LOAD NC MD” softkey
Standard NC MD are loaded

2.3 Press ”CLEAR PLC MD” softkey
PLC MD are cleared

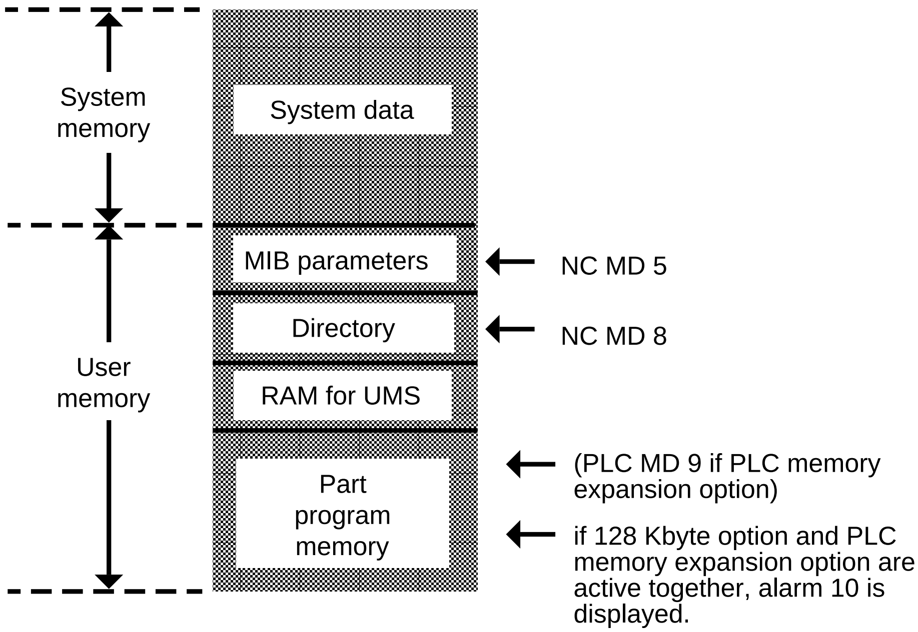
2.4 Press ”LOAD PLC MD” softkey
Standard PLC MD are loaded

3. Recall  key, return to basic display

4. Press "NC DATA" softkey

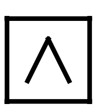
4.1 Press "FORMAT USER M." softkey
The following data are cleared:
- Tool offsets (TO)
- Setting data
- Input buffer (MIB) parameters
- R parameters (local and channel-specific)
- Zero offset (ZO)

The user memory is reformatted, depending on NC MD 5 and 8.
Any modification to these two MD only takes effect after the "FORMAT USER M." and "CLEAR PART PR." softkeys have been activated.



4.2 Press "CLEAR PR." softkey
The part program memory of the NC is cleared.

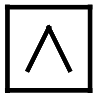
4.3 Press "FORMAT AL-TEXT" softkey
If NC MD 5012 bit 7 is set to "1", the memory is formatted and cleared for the PLC alarm texts (%PCA).

5. Recall  key, return to basic display

6. Press "PLC-INITIAL" softkey

6.1 Press "CLEAR PLC" softkey
The following data are cleared:
– PLC user programs (cyclic and interrupt-driven)
– Input and output images
– NC/PLC interface
– All timers, counters, data blocks
– If PLC memory expansion option is active, the PLC memory (MD9) is set up.

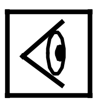
6.2 Press "CLEAR FLAGS"
– All flags are cleared

7. Recall  key, return to basic display

8. Press "SET UP END PW" softkey
The control software triggers POWER ON RESET and then enters NORMAL OPERATION (basic display).

Items 2 and 3 are not necessary if the NC MD and PLC MD are read in via the V.24 (RS 232C) interface. However, items 4 to 8 remain mandatory.

9. Activate option for UMS and configurability (if MADA or PLC program is stored in UMS) and go via POWER ON.

If the machine data are contained in the UMS, they can now be loaded from the UMS into the RAM of the CPU in overall reset mode
(Press  key)

using the "UMS MD" softkey.

The following combinations are possible:

– Clear machine data and then load from the UMS.
– Load standard values (MD) and load MD from the UMS.
(Machine data with value 0 were not stored in the UMS)

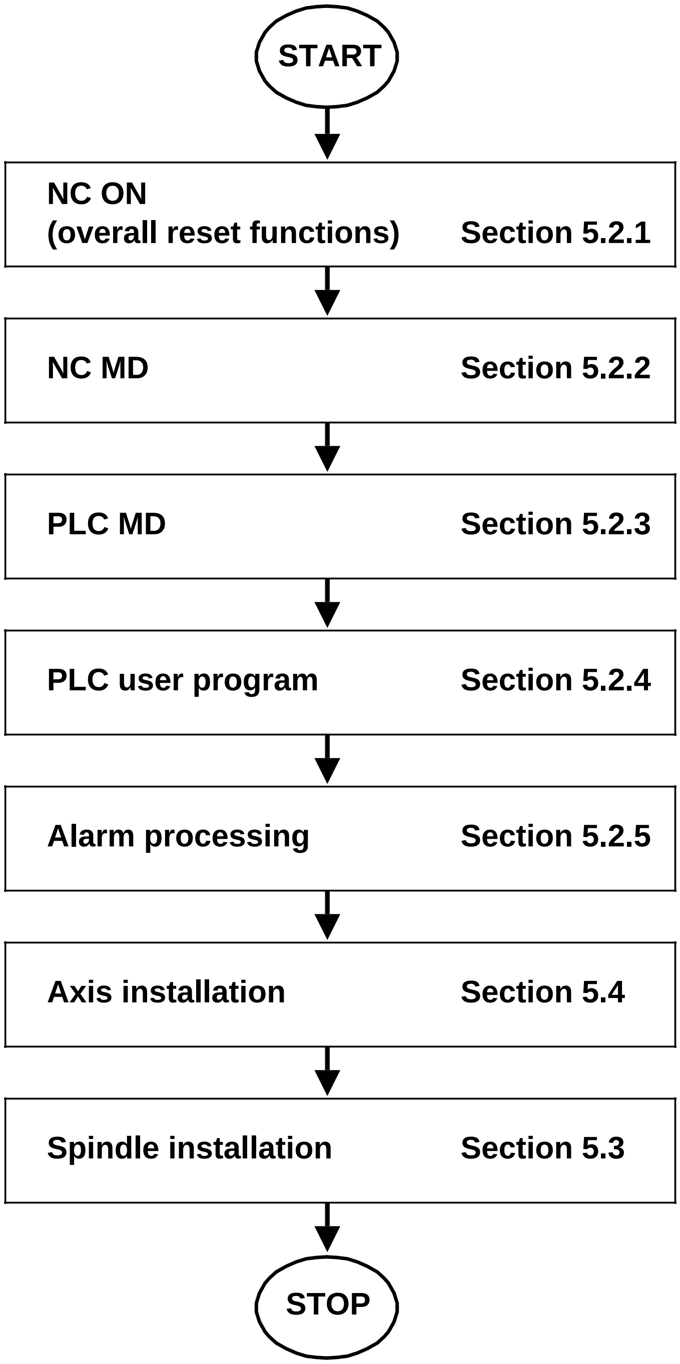
The MDs are stored in the UMS with the following data group identifiers

10 NC machine data
11 PLC machine data
12 Drive machine data

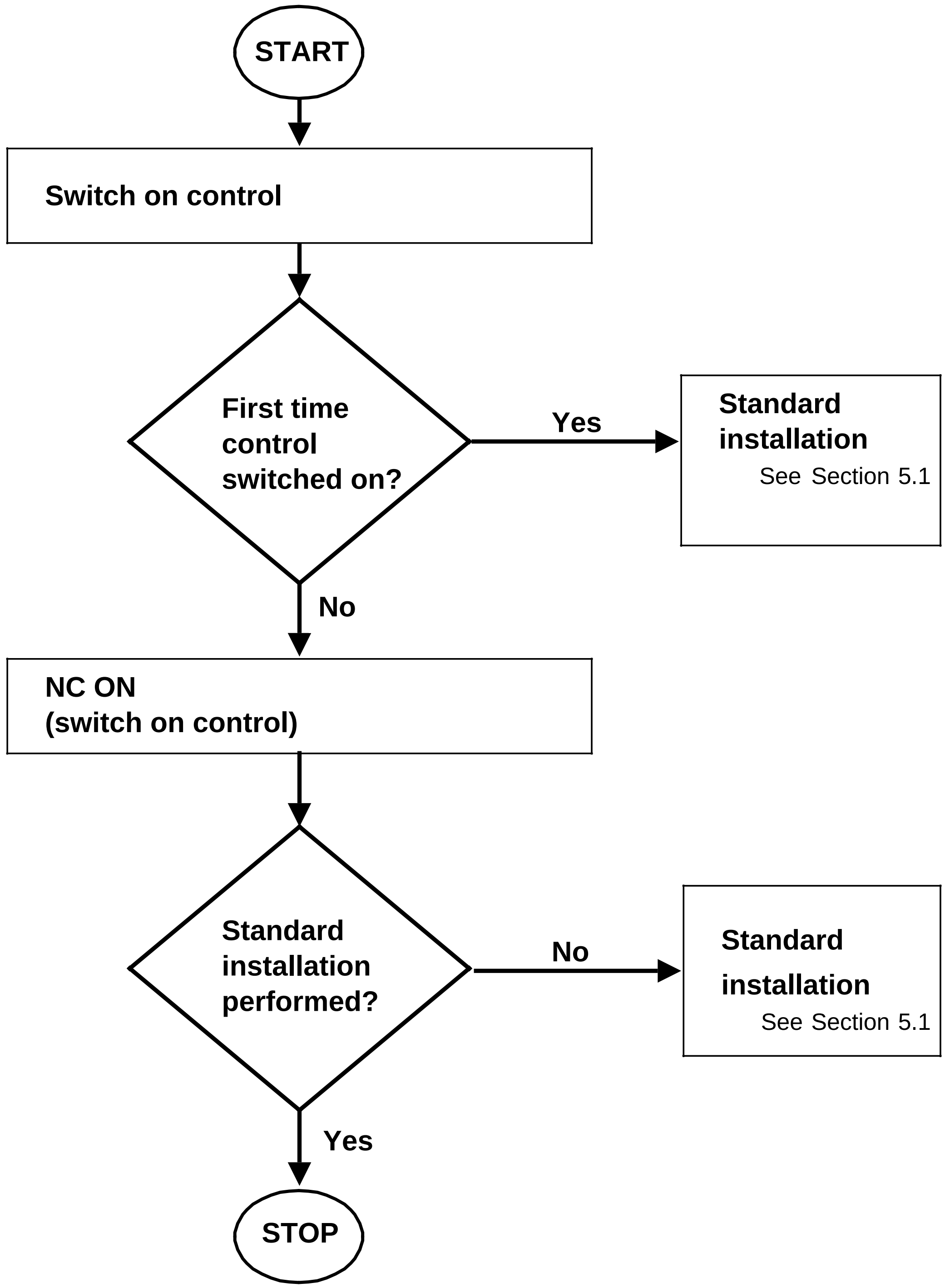
– The PLC program can be stored in "MC-5 code" in the user memory submodule (UMS) of SINUMERIK 810 GA3/820 GA3. This is done either via the WS 800 NC workstation (from V2.0) or via the PG 675/685 configuring package.
– If the PLC program is in the UMS, it can be loaded into the machine data submodule with the "LOAD UMS-PRG " softkey.

The function "LOAD UMS-PRG" can be executed only once in overall reset mode.

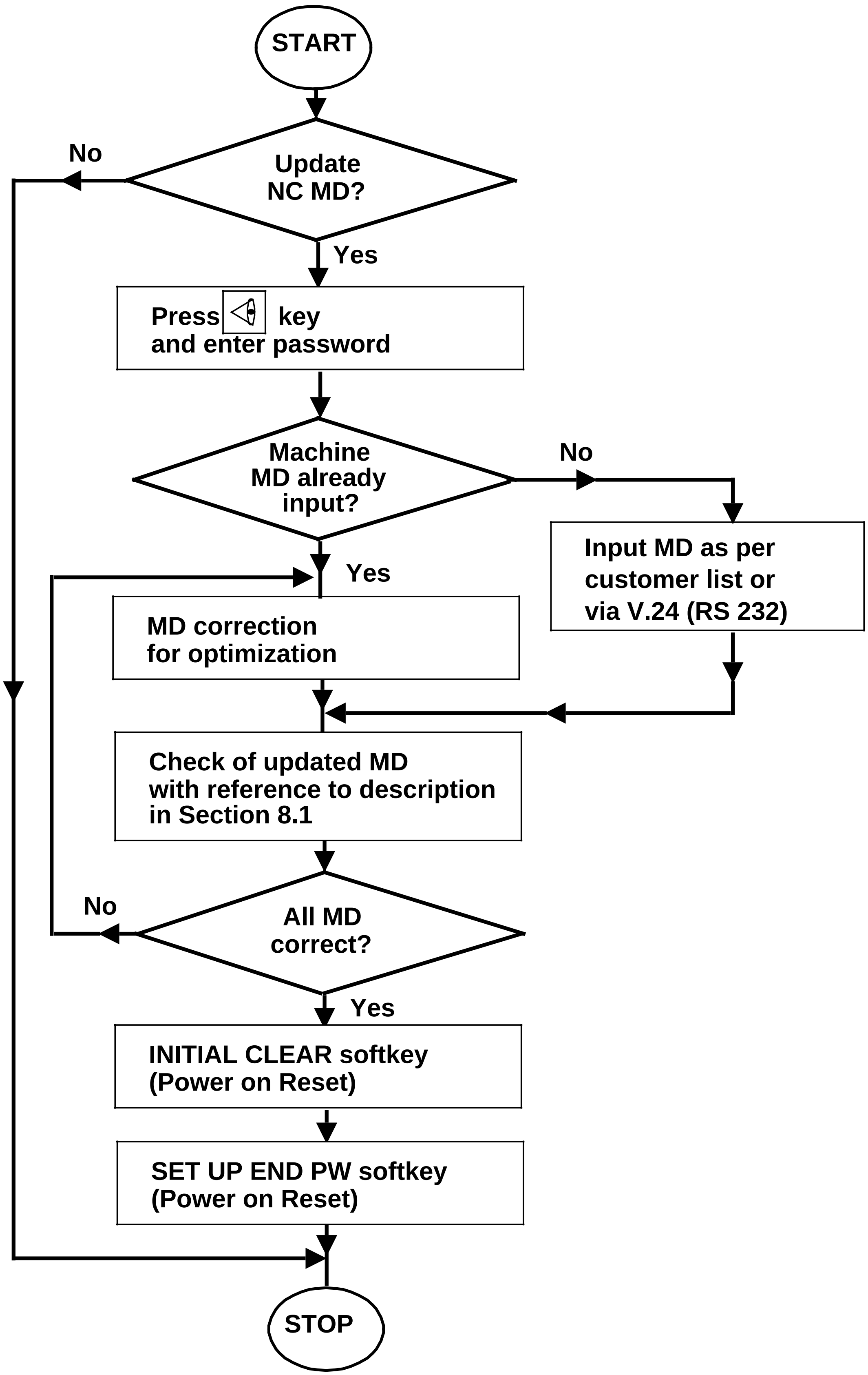
5.2 Standard installation shown as a flowchart



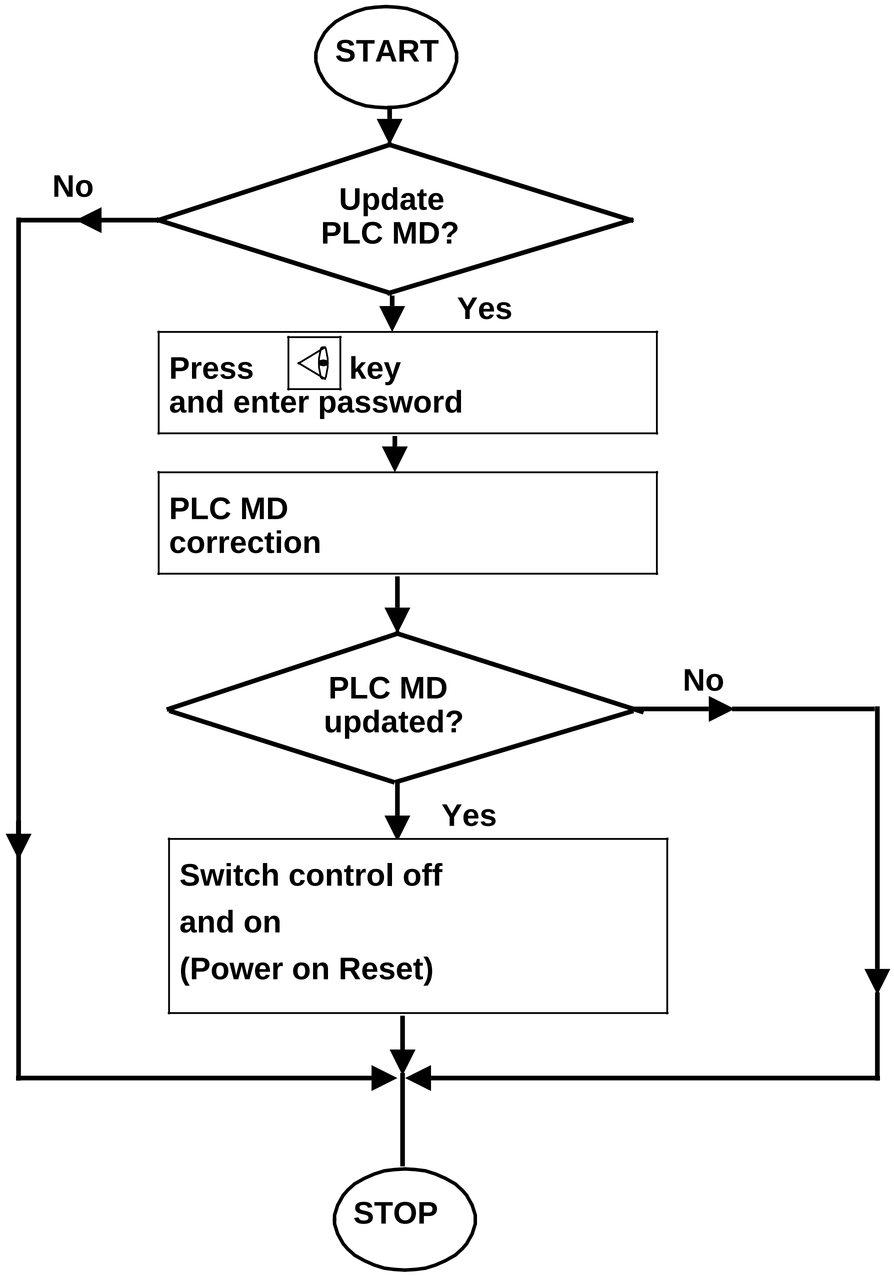
5.2.1 NC ON (overall reset chart and setting standard MD)



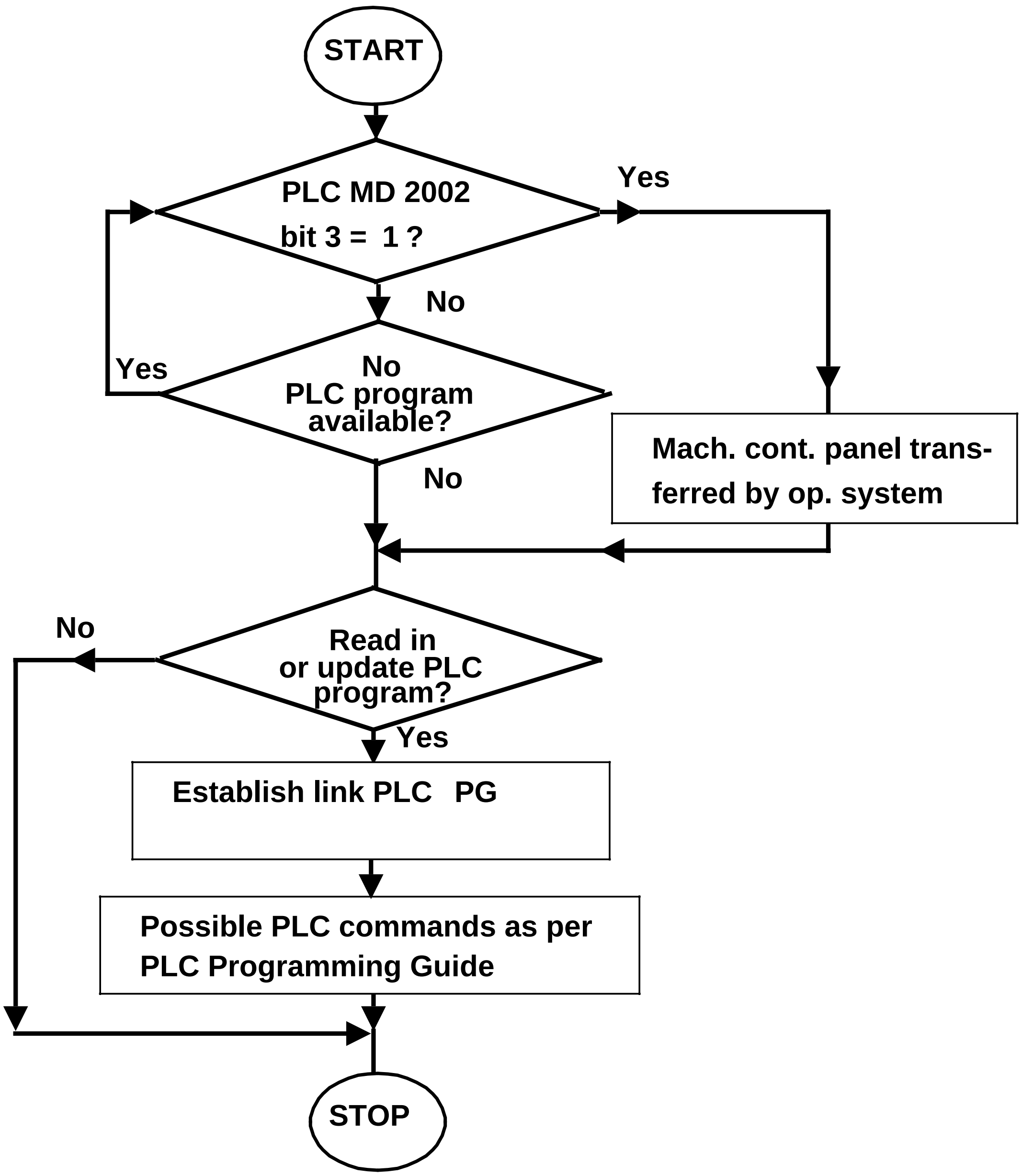
5.2.2 NC machine data



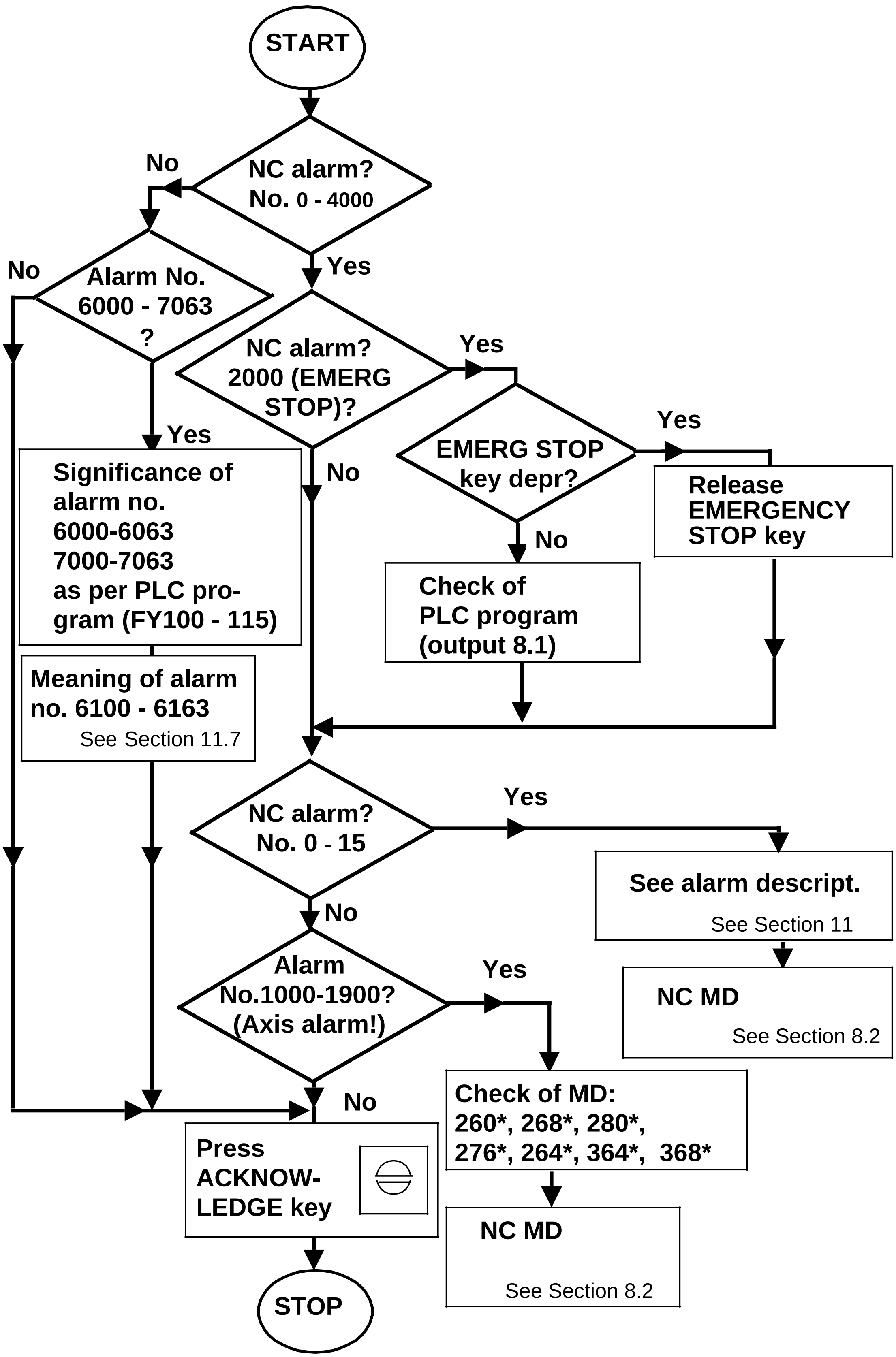
5.2.3 PLC machine data



5.2.4 PLC user program



5.2.5 Alarm processing



5.3 Spindle installation

5.3.1 Requirements

- Installation of the spindle with the setpoint from the battery box and the external enables is completed.
- The setpoint cable (setpoint + servo enable) and the actual value cable is connected.
- The standard machine data is loaded.
- Option F05 (S analog) is available.
- The minimum and maximum speed and the acceleration time constants of every gear stage are known.

For installation of the spindle the following NC MD and PLC   NC signals are important:

NC MD

131 146		Spindle speed override values	
400* 401*		Spindle assignment Spindle drift	
403* : 410*		Max. speed for	1st gear : 8th gear
411* : 418*		Min. speed for	1st gear : 8th gear
419* : 426*		Acceleration time constant	1st gear : 8th gear
444*		Tolerance of the spindle speed	
445*		Tolerance of the max. spindle speed	
446*		Tolerance of zero speed	
447*		Delay for servo enable	
448*		Minimum motor set speed	
451*		Max. spindle speed	
520*	Bit 0	Actual value · 2	
520*	Bit 1	Sign inversion actual value	
520*	Bit 2	Pulse encoder available	
521*	Bit 1	Sign inversion set point	
521*	Bit 7	Spindle available	

*: 0 = Spindle 1, 1 = Spindle 2

Signale PLC   NC

Q 100.7/Q 104.7	Spindle enable
Q 100.6/Q 104.6	Spindle servo enable
Q 100.5/Q 104.5	Define set point ZERO
Q 100.4/Q 104.4	Spindle speed override active
Q 101.7, Q 105.7	Switch on C axis mode (=0)
Q 101.3	Gear stage selection automatic
Q 103.0/Q 107.0	PLC spindle control (= 0)

Procedure:

The following NC machine data must be assigned first:

MD 400*	Value	Spindle assignment
MD 521*	Bit 7	Spindle available

5.3.2 Speed settings for gear stages

NC-MD 403* to 410*	Input max. speed
NC-MD 411* to 418*	Input max. speed

These settings must be made available by the manufacturer.

5.3.3 Acceleration time constants

NC-MD 419* to 426*

These settings must be made available by the manufacturer or must be optimized before installation (see also function description ”M19”).

5.3.4 Spindle setting data

The maximum spindle speed must be entered in setting data 403*.

5.3.5 Testing the spindle in MDA mode

Input MD3/MD4 with an S value:

- Wrong direction of rotation — Change NC MD 521* bit 1
- Does not rotate — Check interface signals

The following interface signals are required:

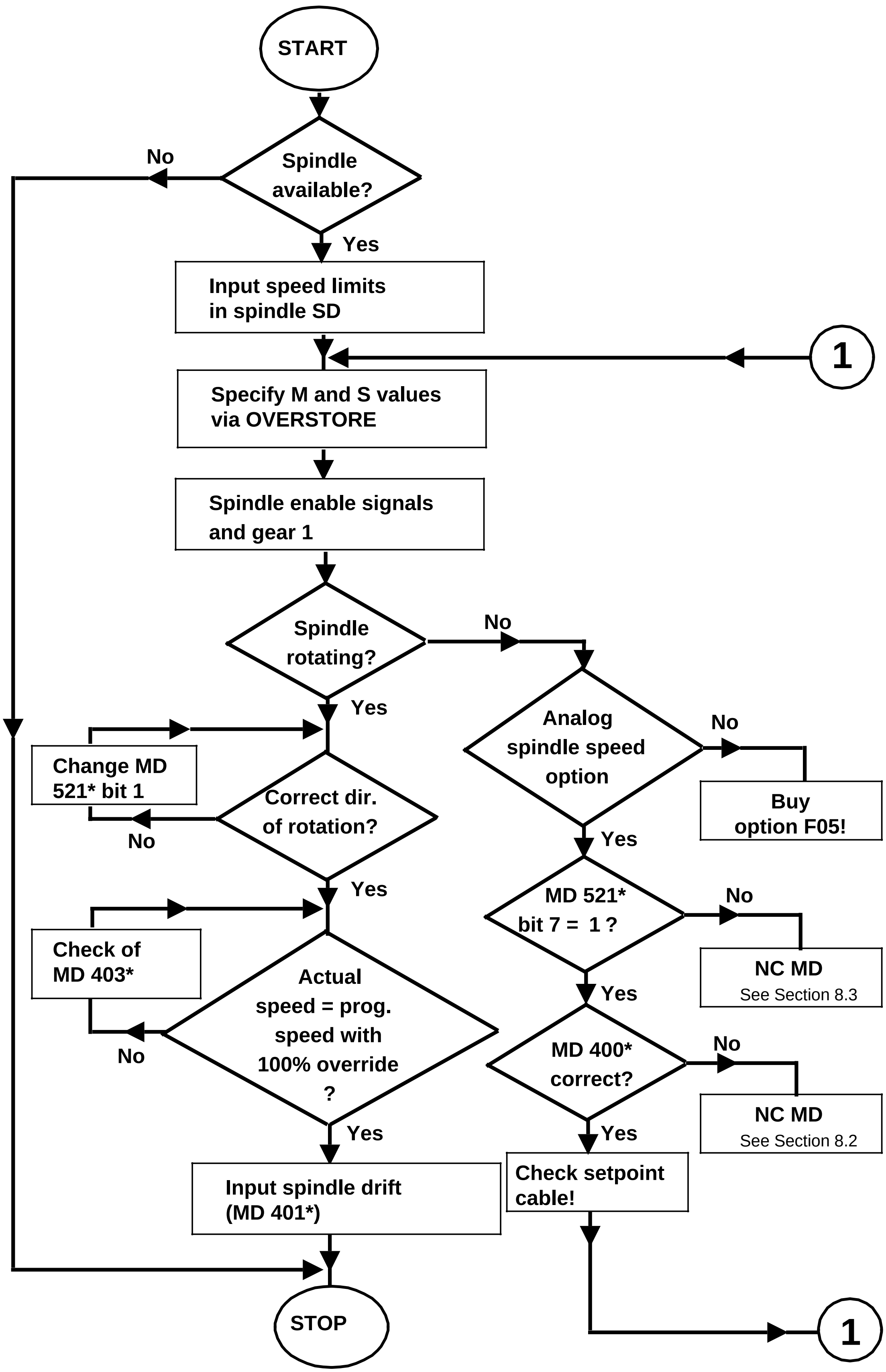
Q 100.7/Q 104.7	Spindle enable
Q 100.6/Q 104.6	Spindle servo enable
Q 100.5/Q 104.5	Do not define set point ZERO
Q 101.3/Q 105.3	Gear stage selection automatic (always ”1” analog!)
Q 100.4/Q 104.4	Spindle speed override active
Q 103.0/Q 107.0	PLC spindle control (= 0)
Q 101.7/Q 105.7	Initiate C axis mode (= 0)

Define S value zero:

- Spindle drifting away — Perform drift compensation (NC MD 401*)

**: 0 = Spindle 1, 1 = Spindle 2*

5.3.6 Spindle installation in sequence



5.4 Axis installation

Axis names

The required axis names must be entered into machine data 5680 to 5684 (for description see Section 8 NC MD 568 *).

The NC programs refer to these axes names.

NC MD	5680	=	1st axis
	5681	=	2nd axis
	5682	=	3rd axis
	5683	=	4th axis
	5684	=	5th axis
	5685	=	6th axis
	5686	=	7th axis

General notes:

Before beginning NC axes installation the axes should already have run error-free with the battery box as the setpoint encoder. The following measures must already have been taken:

- Adaptation of the controller module to the motor (max. current, current curve set depending on the speed).
- Tacho-generator adaptation: At 95 % of the maximim setpoint voltage the maximum speed must have been reached.
- Optimization of the speed and current controller must have been performed (no overshoot or overdamping).
- Remove setpoint connector from the NC.
- Standard MD must be loaded.
- Leave actuator switched off.

The ”servo enable” signal which was brought from the NC in the set point connector for every axis to the drive actuator must be used at the drive actuator.

5.4.1 Feedback polarity of the feed axes - checking and adjustment

It is imperative that the position feedback polarity and speed feedback polarity should be checked **before** closing the control loop because an incorrect polarity results in uncontrolled axis movements at maximum speed.

The following must therefore be clarified prior to starting work:

- Traverse direction of feed axes (according to customer or to ISO)
- Polarity of the set speed voltage at the control unit for axis movements in the positive direction.

Example:

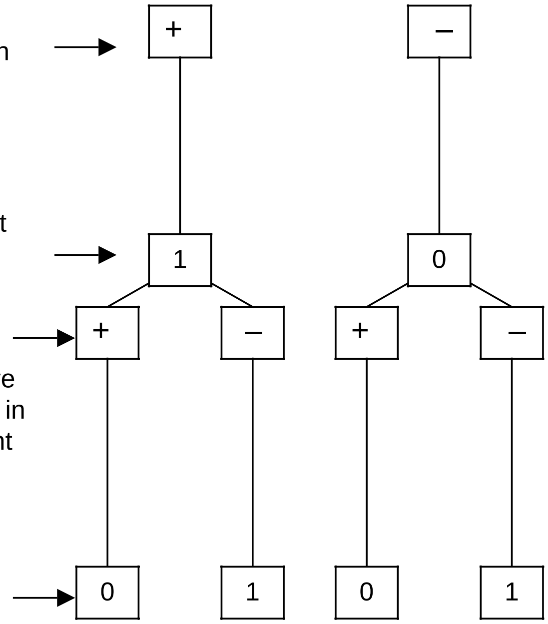
With which polarity of the set speed voltage on the battery box does the axis move in the positive direction?

Set machine data bits for sign inversion for set speed (MD 564* bit 1).

Test the position feedback polarity:

Move the feed axis mechanically in the positive direction. Observe the direction of the change in actual value on the display beneath the current actual value.

Sign inversion for actual value difference (MD 564* bit 2).



5.4.2 Position control resolution, input resolution

Before closing the position control loops (plug in the setpoint's connector) of the axes the input resolution (NC ND bits 5002.4 to 5002.6), the display resolution and the position control resolution (NC ND bits 5584*) must be entered (refer to NC ND 5002 and 584* for permissible bits combinations).

The axis-specific max. speed (NC MD 280*) and the position control resolution are related in the following way:

Max. axis speed	Position control resolution
2.4 m/min	$0.5 \cdot 10^{-4}$ mm
24 m/min	$0.5 \cdot 10^{-3}$ mm
240 m/min	$0.5 \cdot 10^{-2}$ mm

Axes which are not involved in a 2D-/3D interpolation can also be assigned as follows

Max. axis speed	Position control resolution
4.5 m/min	$0.5 \cdot 10^{-4}$ mm
45 m/min	$0.5 \cdot 10^{-3}$ mm
450 m/min	$0.5 \cdot 10^{-2}$ mm

Note:

Because the position control resolution applies to all axes it must be set for the axis with the greatest maximum speed.

5.4.3 Maximum speed of the axes

The axis-specific maximum speeds must be entered in the NC machine data 2800 to 2806. The axis-specific maximum speeds are defined by the manufacturer according to the requirements and the mechanical design (see NC MD 280*).

5.4.4 Definition of the maximum setpoint

The axis-specific maximum speeds (NC MD 280*) required by the customer are assigned to a set speed voltage (NC MD 268*).

Note that an additional control margin of approximately 5% is required.

The power limits are set by the DAC of the setpoint (10V) or by the control unit of the drive.

The setpoint limit is set via NC MD 268*. In this way the output voltage of the NC is limited. In operation, the limit entered in NC MD 268* must not be reached.

The maximum admissible setpoint is 10V.

Conversion into input data 10V ≙ approx. 8192 units

The maximum setpoint must be input according to the maximum permissible input voltage of the drive control unit.

a) Maximum permissible input voltage of the drive control unit 10V:

Input in NC MD 268* : 8192 units = 10V

The maximum axis speed is already reached at 9.5V because of the control margin of 5%.

b) The drive control unit works with the maximum set speed voltage of 10V, e.g. 5V:

Input in NC MD 268* : $\frac{8192}{10\text{ V}} \cdot 5\text{V} = 4096$

- Max. input voltage of the actuator: 5V
- Input in NC MD 268*: 4096
- Tacho-generator compensation: At 4.5V the maximum speed should have been reached.

The setpoint should generally be set as high as possible because the higher setpoint voltage achieves better control response.

5.4.5 Variable incremental weighting

- Input values for NC MD 364* (number of pulses for variable increment weighting) and NC MD 368* (traverse path for variable increment) as described in the section on ”NC machine data”.
- Subsequent check of the values:
  Traverse 1000 increments, for example for JOG-INC-VAR mode: calculate the path the depending on the set input resolution (1 increment = 1 unit (IS)) and compare with the reading on a dial gauge.

5.4.6 Closing of position control loop

With the control disconnected, insert the setpoint connector and cancel any interlocks for this axis (fuses, servo disable). Lock other axes, switch on the control, switch on actuator.

Press "EMERGENCY STOP" immediately if the feed axis

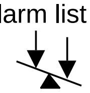
performs an uncontrolled movement

in order to avoid danger to man and machine.

Cause of uncontrolled movement	Characteristic feature
Incorrect polarity of position control loop or speed control loop (NC machine data bits incorrect)	Axis moves at maximum speed
Position control loop not closed	Axis moves at slow, constant speed. Measuring equipment does not follow axis movement (e. g. coupling loose) Short-circuit to frame, open circuit or cable short-circuit causes measuring-circuit monitor to respond
Setpoint not at speed controller	Axis moves at slow, constant speed (drift)
Control loop fault • Open circuit in tacho-generator feedback loop • Incorrect polarity of tacho-generator feedback loop • Incorrect optimization • Servo gain factor excessive	Oscillating and severe reciprocation of axis

5.4.7 Traversing the axes in JOG mode

Setpoint cable plugged in, feedback polarity and pulse weighting correct. Traverse the axis at low speed using the direction keys.

- If the "Feed hold" message appears, check the interface signals.
  The following interface signals are required:
  - feed enables (for each axis and overall)
  - no axis disable
  - no follow-up mode
  - servo enable
  - feed override not at zero
- If alarms appear see alarm list
- +/- character in motion ()
  LED field remains lit perform drift compensation (see NC MD 272*)

5.4.8 Multgain NC MD 260*

A multgain factor must be entered in NC MD 260* to calculate the set speed. It is thus posible to operate axes at different maximum speeds with full utilization of the setpoint input.

Axes traversing in continuous-path mode must have the same servo gain factor (K_v factor). For this reason, the multgain factor must be determined for each axis using the following equation:

Metric measuring system:

$$\text{Multgain} = \frac{3 \cdot 10^7}{V_{max}\ [1000\ \text{units(MS)/min}]} \cdot \frac{U_{max}\ [V]}{10V}$$

Example: Maximum axis speed V_{max} = 20 m/min = > position control resolution 0.5 . 10^{-3} mm = > 1 unit (MS) = 2 · 0.5 · 10^{-3} mm = > V_{max} = 20000 (1000 units/min) = set point voltage at V_{max} = **9.5V**

$$\text{Multgain} = \frac{3 \cdot 10^7}{20000\ [1000\ \text{units(MS)/min}]} \cdot \frac{9.5\ [V]}{10V} = \mathbf{1425} \left[\frac{\text{min}}{1000\ \text{units(MS)}}\right]$$

Note:

If an inch input system (IS) is used then the max. speed (MD 280*) must be converted to mm/min and entered in the formula as V_{max}.

Inch measuring system:

Measuring system weighting 1/2 · 10^{-4} inch:

$$\text{Multgain} = \frac{3 \cdot 10^7}{V_{max} \cdot 0.1\ [1000\ \text{units(MS)/min}]} \cdot \frac{U_{max}\ [V]}{10V}$$

Measuring system weighting 2 · 10^{-5} inch:

$$\text{Multgain} = \frac{1.2 \cdot 10^7}{V_{max} \cdot 0.1\ [1000\ \text{units(MS)/min}]} \cdot \frac{U_{max}\ [V]}{10V}$$

Note:

If a metric input system (IS) is used then the max. speed (MD 280*) must be converted to inch/min and entered in the formula as V_{max}.

Note:

The standard encoder pitch in inch measuring systems is 0.1 inch. With a measuring system weighting of $1/2 \cdot 10^{-4}$ inch, this pitch must be increased to 0.2 inch by means of appropriate gearing with respect to the encoder or by setting the machine data bit ”Divide actual part position by 2”.

5.4.9 Servo gain factor K_V factor

A high servo gain value is required to ensure that only minor contour deviations occur in continuous-path operation.

However, an excessive (K_V factor) value results in instability, overshoot and, possibly, impermissible machine loading.

The maximum allowable servo gain factor depends on the the following:

- Design of drive system
  (rise time, acceleration and braking power)
- Quality of machine.

The servo gain factor (K_V factor) is defined as follows:

$$K_V = \frac{\text{Speed}}{\text{Following error}} \qquad \frac{[\text{m/min}]}{[\text{mm}]}$$

$\frac{\text{m/min}}{\text{mm}}$ is the unit of the servo gain factor (K_V factor) to VDI standard

If an empirical value for the servo gain factor (K_V factor) is known for the machine, this value is set and a check is performed for overshoot or instability.

A precondition for correct setting of the servo gain factor is good speed controller optimization.

Setting of servo gain factor

Reduce acceleration (NC MD 276*). The overshoot behaviour is the decisive factor in assessing the maximum servo gain factor. Consequently, the acceleration setting must be such that the drive remains below its current limit.

Should the drive reach an acceleration value of 1 m/s^2, half the value should be selected for reasons of safety.

0.5 m/s^2 $\hat{=}$ input 50

The closed-loop gain is entered under NC MD 252* using the following conversion formula:

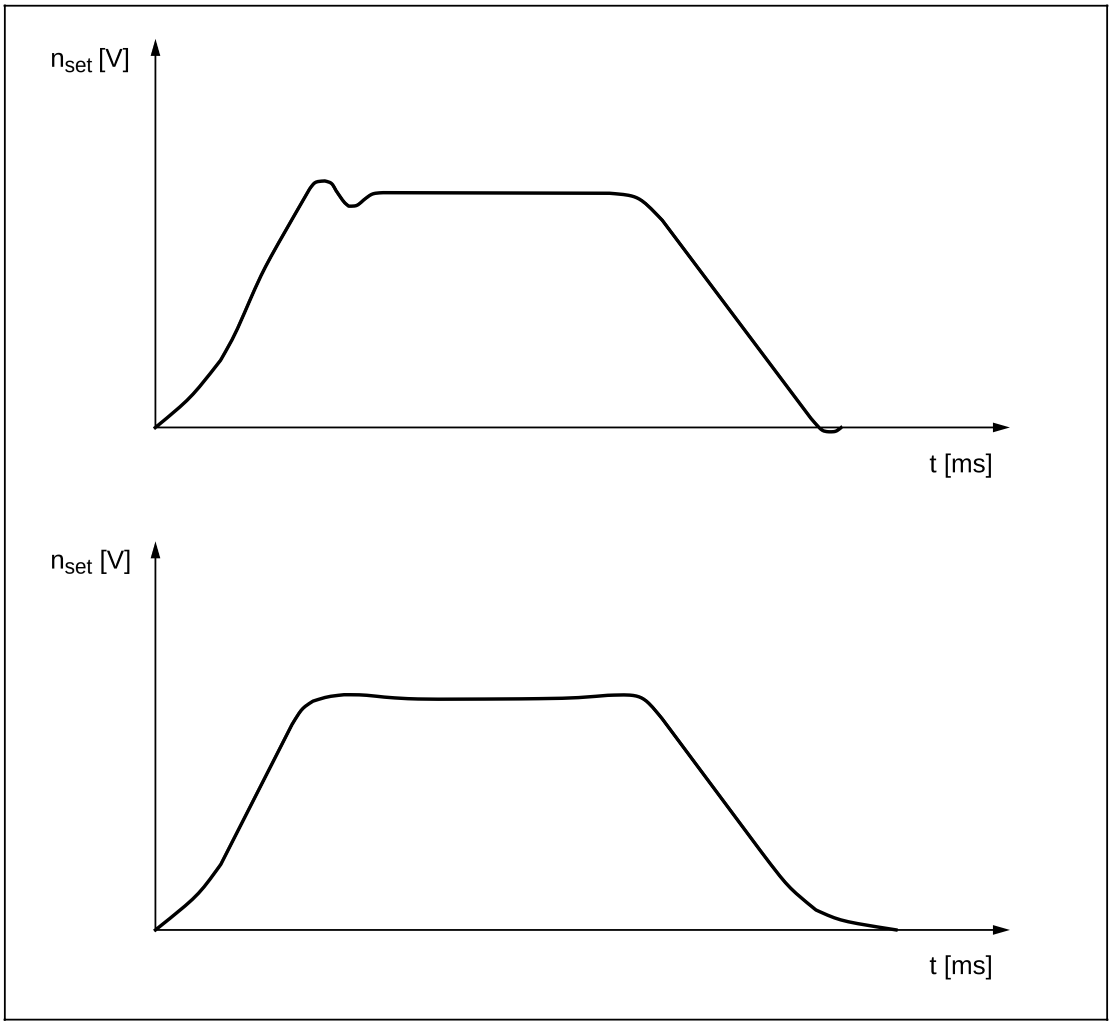
$$K_v\,(0.01\ s^{-1}) = \frac{5000}{3} \cdot K_v \left[\frac{m/min}{mm}\right]$$

$$= 1666 \cdot K_v \left[\frac{m/min}{mm}\right]$$

For K_v factor 1, therefore, the numerical value 1666 is entered.

To assess whether the approach behaviour is correct and the maximum value correctly set, you take the dynamically most unfavourable axis contributing to continuous-path operation.

The setpoint voltage n_{set} to the speed controller is measured with an oscillomink or storage oscilloscope. Various speeds are used to traverse.



In particular, deceleration is to be observed with higher voltage gain on the oscilloscope or oscillomink.

Overshoot may also occur on account of the following factors:

- Acceleration excessive (current limit reached)
- Speed loop rise time excessive
- Fault in speed controller (re-optimization may be necessary)
- Mechanical backlash
- Canting of mechanical components
- Load fluctuations (perpendicular axis)

For safety reasons, a servo gain factor should be selected which is at least 10% smaller than the maximum possible value.

Axes working together in continuous-path operation must have the same servo gain factor.

Check of servo gain factor

The magnitude of the following error is shown by the service display of the individual axes. If the drift has been compensated, the values displayed for the positive and negative traversing directions are identical.

Finally, the servo gain factor entered for all axes must be checked during operation via the following error display.

Precise continuous-path operation requires equal dynamic axis behaviour, i. e. the same following error must occur at the same speed.

Any differences must be compensated in terms of multgain or at the actual-speed potentiometer.

5.4.10 Acceleration

The axes are accelerated and decelerated with the accelerations entered in NC MD 276*

$b \cdot 10^{-2}$ [m/s^2]

Acceleration to relevant speed and positioning are thus possible with accuracy and speed and with minimum strain on the machine.

The customer should be asked for which maximum axis acceleration the machine is suitable. This value (assuming the drive is not overloaded) is entered under NC MD 276*.

These values are normally situated between

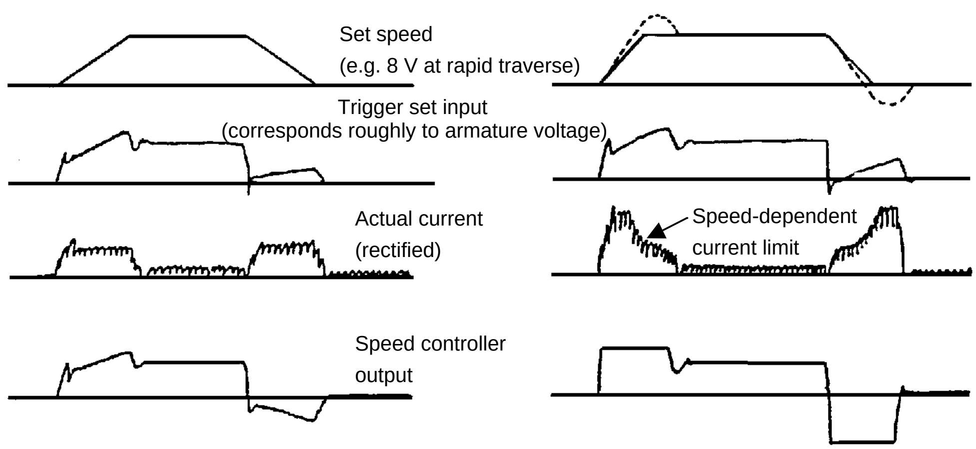
0.3 m/s^2 and 2 m/s^2

Checking or determining the acceleration values:

Setting:	NC MD 276*
Condition:	Overshoot-free acceleration or positioning at rapid traverse rate (acceleration stop limit) under maximum load conditions (heavy workpieces on table)
Measuring equipment:	Recorder or storage oscilloscope
Measuring point:	Set speed and possibly actual current and speed controller output

Once the acceleration has been set, the rapid traverse rate is selected and the actual currents and possibly speed controller output are simultaneously recorded. It can thus be seen whether or not the current limit has been reached. The drive may briefly reach the current limit. However, this must only occur in the rapid traverse range. For a period prior to positioning, the drive must be in the speed control cycle once more, otherwise the axis overshoots its position.

Examples: 6 pulse circulating current feed drive with current limiter.



Example 1:

Appropriate acceleration

Example 2:

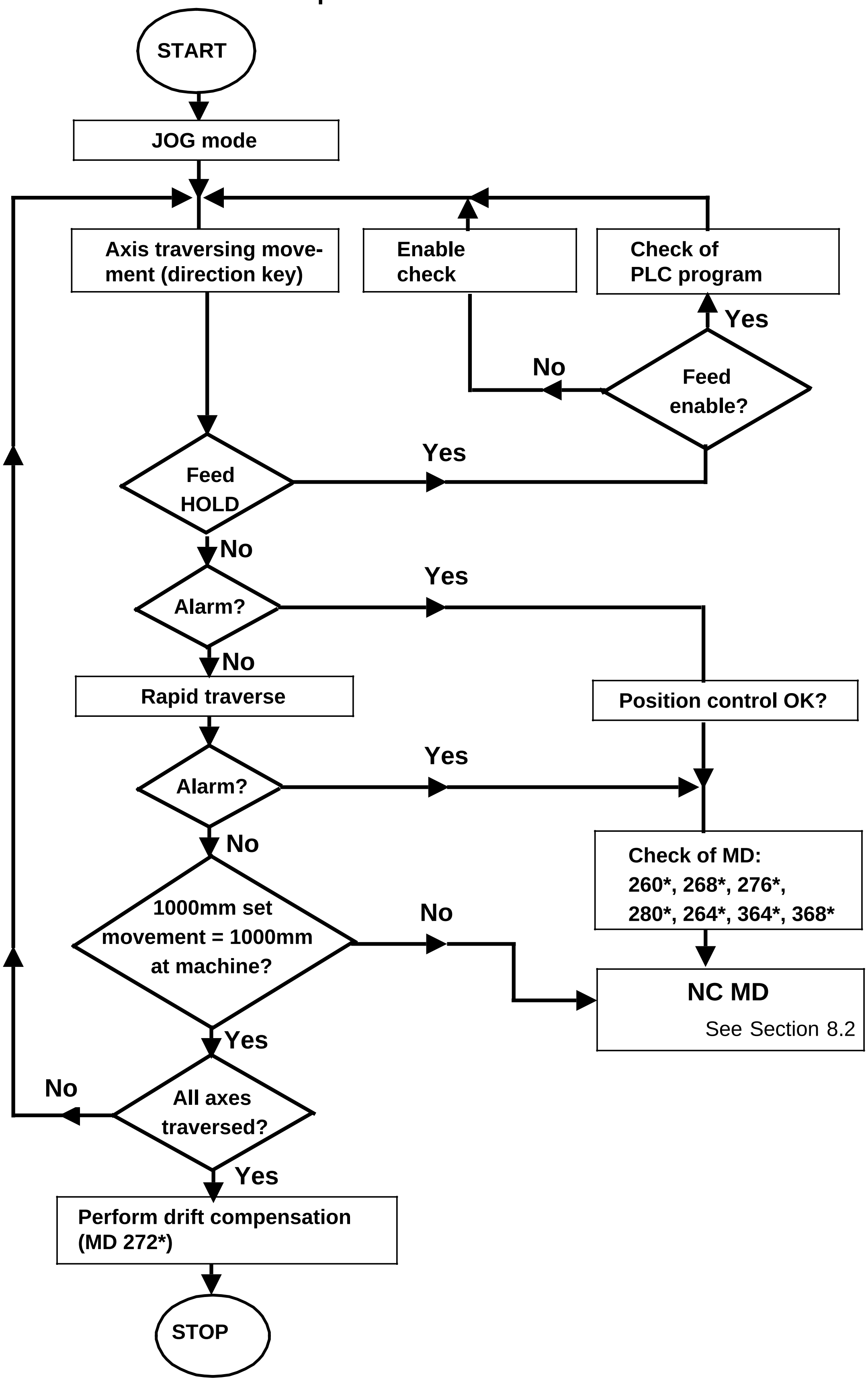
Acceleration too high,
current limit is reached.
---- With the current loop the set point is overshot at the top and the bottom because of the increased following error.

Slight changes in load (sluggishness, effect of lubrication) must not result immediately in the current limit being reached. Consequently, at least a 10 % lower acceleration value should be entered.

At the customer's request, acceleration may be further reduced to minimize the load on the mechanical components.

The axes can be given different acceleration values.

5.4.11 Axis installation sequence



5.5 Reading in machine data from the UMS

The machine data of SINUMERIK 810/820 can be read out via the V.24 (RS 232) and entered in the UMS at the NC workstation. The machine data are given the following data group identifiers in the UMS:

10 NC machine data
11 PLC machine data
12 drive machine data

The machine data are stored in fixed format according to the number of machine data (1 word fixed point). The value is stored in BCD 9 S format according to the MD No. (1 word fixed point). Machine data with the value 0 are not stored in the UMS.

The following sequence of operations is used to load the machine data from the UMS.

1. 1st installation (standard installation).
2. Set option for UMS and configurability.
3. Activate Power On once in normal mode.
4. Select overall reset mode.
5. Clear machine data.
6. Load standard values, if applicable.
7. Transfer machine data from the UMS to the main store using the ”LOAD MD-UMS” softkey.
8. Machine data with setpoint 0 must be deleted manually if the contents do not equal 0 (e.g. after loading the standard values).

6 PLC Description

6.1 Technical specification

- 128 bytes inputs
  including max. 32 bytes free for user
- 128 bytes outputs
  including 16 bytes free for user
- 256 bytes flags
  including 63 bytes free for user
  24 bytes reserved for standard FBs
  32 bytes transfer flags
- 32 timers 0 ... 31
  all free for user
- 32 counters 0 ... 31
  all free for user
- 16 Kbytes for program blocks (including pre-header)
- 6 Kbytes or data blocks (including pre-header)
- 32 Kbytes PLC memory expansion (option)
- PLC cycle time=12 x sampling time for position controller
  dependent on NC MD 155 and program length
  Example : NC MD 155=2 servo sampling rate of 5.0 ms
  PLC cycle time=12 x 5.0=60 ms
- 5.0 ms response time for PLC interrupt control
  dependent on NC MD 155

6.2 PLC MD, PLC MD bits

6.2.1 General

Overview of PLC MD:

PLC MD	0	...	9	System data
PLC MD	1000	...	1007	User MD
PLC MD	2000	...	2005	System bits
PLC MD	3000	...	3003	User bits

All PLC machine data are only active after

PLC cold restart (Power On).

The programming language used is STEP 5. A detailed list of the possible statements is contained in the programming instructions ”Programming PLC SINUMERIK 810, GA3/ SINUMERIK 820 GA3”.

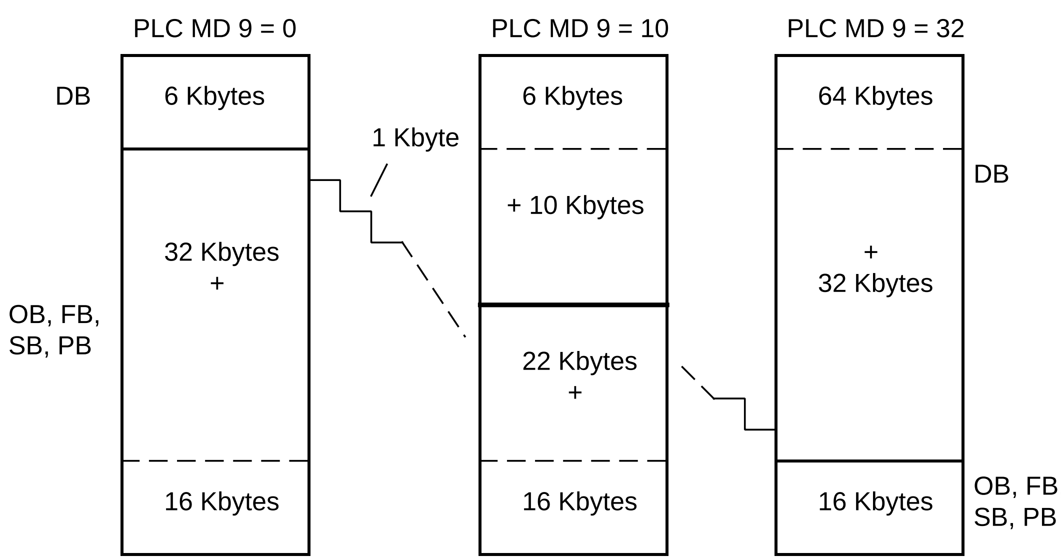
PLC memory overview
without PLC memory expansion:

3 K words data blocks
8 K words OB 1, OB 2, OB 20 PB, SB, FB

PLC memory overview

with PLC memory expansion:

The PLC memory is increased in size by 32 Kbytes. The additional memory is split up over the PLC areas DB or OB, SB, FB and PB with the help of PLC MD 9.

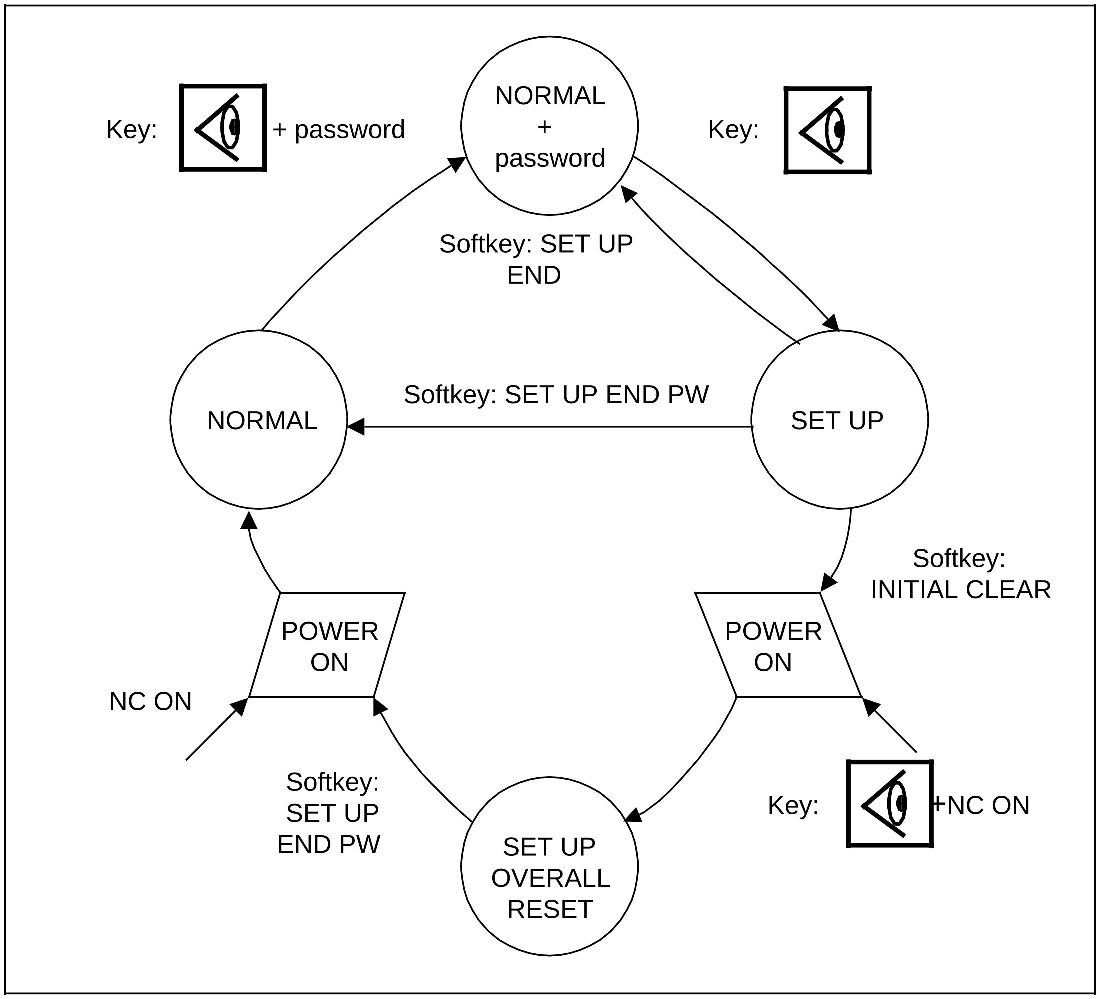


The PLC memory expansion precludes the option ”Part program memory expansion to 128 K”.

6.2.2 MODE switching (types of operation)

MODE (type of operation)

Power On routines



SET UP END PW = end of installation

NORMAL:

- All functions as per Operator's Guide
- Reading of MD
- PLC-STATUS (read only)

NORMAL+password:

- All functions as per Operator's Guide
- Reading and writing of MD
- PLC-STATUS (read and write)

SET UP:

- Clearing of password
- Branch to ”SET UP overall reset” mode

SET UP OVERALL RESET:

In this MODE the machine cannot be operated.

The following functions are available:

- Overall reset (cancel function)
- Load standard MD (input function)
- RS232C (V.24) activation
- PLC cannot be started via PG
- Read in/out PLC programs with S-5 operating system or C/PM operating system (%PCP)
- Read in/out PLC alarm texts (% PCA)

6.2.3 MD description

PLC-MD 0	No. for interrupt-processing input byte

If interrupt-driven program processing has been activated (PLC MD 2002 bit 0), the number of the input byte with which interrupt processing is initiated (OB2) is defined with PLC MD0. Each edge change at one of these inputs results in the interrupt handler stored in OB2 being called.

If interrupt processing is not selected, this input byte behaves as a standard I/O device.

Standard	IB: 7
Max. permissible	IB: 63

Note:

The interrupt input byte may be assigned to both the centralized and decentralized I/O device. It is checked for a change every 5.0 ms (depending on NC MD 155).

PLC-MD 1	Max. interpreter run time 0B1+0B2

Max. load on CPU in % not to be exceeded by the cyclic and interrupt-driven PLC user program. If the MC5 exceeds this specified percentage CPU load, the user can respond specifically through PLC MD 2003 bit 1.

- PLC MD 2003 bit 1=0
  PLC does **not** move to Stop when the interpreter run time exceeds PLC MD 1. Only PLC alarm 6159 (run time exceeded S-5 program) is then triggered.
- PLC MD 2003 bit 1=1
  PLC moves to Stop when the interpreter run time exceeds PLC MD 1. In addition, PLC alarm 6159 is triggered.

Function of interpreter :

During control read-in, the PLC program is not translated into an Assembler 80186 program but is stored in MC5 code (machine code of STEP 5 language). The MC5 interpreter translates each MC5 instruction into an operational Assembler 80186 program and thus processes STEP 5 statements.

Calculation of the maximum permissible interpreter run time 0B1+0B2 is dependent on PLC MD 2003 bit 6 (segmentation of S-5 program processing).

a) PLC MD 2003 bit 6=0 (no segmentation)

The percentage interpreter run time refers to an interval of 12 times the position controller sampling time (NC MD 155), which also corresponds to the PLC cycle time.

Example:

NC MD 155=2	Position controller sampling time		=	5.0 ms			
		Cycle time	=	12 x 5.0	=	60	ms
	Assuming:	PLC MD 1	=	15%			
	Max. permissible interpreter run time 0.15 x 60 ms				=	9.0	ms

i.e. max. permissible interpreter run time 0B1+0B2 within a PLC cycle, in this case 60 ms, must not exceed 15%=9.0 ms.

b) PLC MD 2003 bit 6=1 (segmentation of S-5 program)

The percentage interpreter run time refers to an interval of 4 times the sampling time of the position controller. The PLC cycle time is now dependent on the S-5 program length.

Example:

NC MD 155=2	Position controller sampling time		=	5.0 ms			
		Time interval	=	4 x 5.0	=	20	ms
	Assuming:	PLC MD 1	=	15%			
	Max. permissible interpreter run time 0.15 x 20 ms				=	3.0	ms

Standard: 15%
Max. value: 20%

PLC MD 2	PLC call OB1

This MD is used to select the time reference of the OB1 call. The time of the Ipo cycle (4 times the value of NC MD 155) is used as the time reference unit.

The following values are permissible:

1≙1 Ipo cycle
2≙2 Ipo cycles
0, 3, 4≙3 Ipo cycles

Example:

NC MD 155=2
PLC MD 2=0

For the Ipo cycle, this results in 20 ms and for the PLC call: 3 x 20 ms=60 ms.

PLC MD 3 — Max. permissible interpreter run time OB2

Max. interpreter run time in µs required by the PLC interrupt program within 4 times the sampling time of the position controller (NC MD 155). If this run time is exceeded (time-out), the user may respond specifically through PLC MD 2003 bit 0.

- PLC MD 2003 bit 1=0
  No PLC stop when the interpreter run time is exceeded (time-out). Only PLC alarm 6160 (run time exceeded OB2) is triggered.
- PLC MD 2003 bit 1=1
  PLC moves to Stop when the interpreter run time is exceeded (time-out). In addition, PLC alarm 6160 is triggered.

Example: 4 x sampling time = 4 x 5.0 = 20 ms
Max. perm. interpreter run time = 2000 µs
Run time of OB2 with 4 alarms = 500 µs

The sum of the interpreter run times within these 20 ms must not exceed 2000 µs (irrespective of the number of OB2 calls).

Standard value: 2000 µs
Max. value: 2500 µs

PLC MD 5 — Cycle monitoring

Max. time in ms required by the PLC user program (OB1+OB2). If this time is exceeded (time-out), the PLC assumes the Stop state. PLC alarm 6161 is then activated. Without segmentation of the PLC program processing, no effect (PLC MD 2002.6).

Standard value: 70 ms
Max. value: 320 ms

PLC MD 6 — No. of last enabled S-5 timer

The user has available max. 32 timers (0 - 31) in his S-5 program. These timers must be continuously updated, i.e. processed, by the system program, even if they are not programmed by the user. This gives rise to an unnecessary load on the CPU as a result of the PLC system program. PLC MD 6 allows the user to specify the timers from T0 to T31 enabled for his program.

Example: PLC MD 6 = 10
T0 to T10 = 11 timers enabled

Standard value: 15
Max. value: 31

PLC MD 8 — No. of the interface for DB 37

With this PLC MD a serial interface is assigned to the PLC. This interface can be controlled via the DB37 (optional).

With SINUMERIK 810 GA3/820 GA3
only one interface can be active at a time.

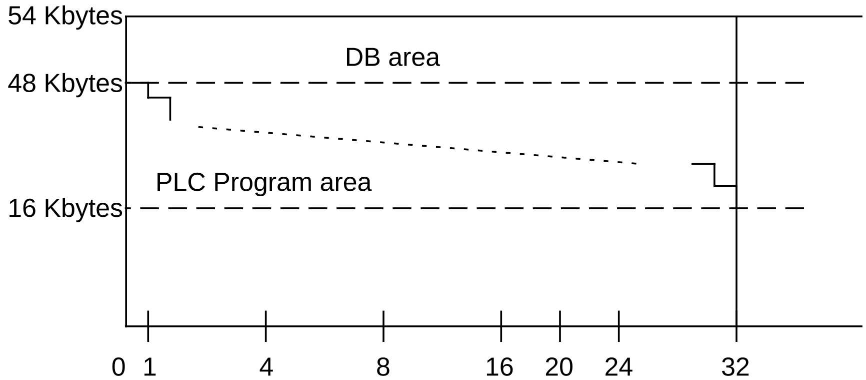
PLC MD 9 — Selection of Kbytes for DB area

With this PLC MD, the number of additional Kbytes for the DB area is specified when the PLC memory expansion option is active.

Example:

PLC MD 9	=	0:	DB area has 6 Kbytes of memory available PLC program area has 16+32 Kbytes of memory available
	=	32:	DB area has 6+32 Kbytes of memory available PLC program area has 16 Kbytes of memory available

Available PLC memory



Standard value: 8
Max. value: 32

PLC MD 1000 ... 1007	**User MD words**

These MD words may be used freely by the user. With each COLD RESTART of the PLC they are transferred to flag bytes 120-135, where they may be processed by the PLC program (e.g. to supply timers).

Corresponding PLC MD: 2000

PLC MD	Flag word
1000	FW 120 (FY 120 and FY 121)
1001	FW 122 (FY 122 and FY 123)
.	.
.	.
.	.
1007	FW 134 (FY 134 and FY 135)

Max. value: 0 ... 65535 for fixed-point number
0 ... 9999 for BCD number

Bits for PLC MD 2000	PLC user MD in BCD code

Bit:	7	6	5	4	3	2	1	0
	Word 8 in BCD	Word 7 in BCD	Word 6 in BCD	Word 5 in BCD	Word 4 in BCD	Word 3 in BCD	Word 2 in BCD	Word 1 in BCD

PLC user MD 1000-1007 may be stored not only as a fixed-point number (value range: 0-65535) but also as a BCD number (value range: 0-9999).

The distinction is based on the corresponding bit in PLC MD 2000.

Example: PLC MD 1006=2048

MD 2000.6 =	”0”:	FW 132	KH=0800	(word 7 stored as fixed-point number)
MD 2000.6 =	”1”:	FW 132	KH=2048	(word 7 stored as BCD number)

Bits for PLC MD 2001	NC-PLC data transmission in BCD code

	NC → PLC							
Bit:	7	6	5	4	3	2	1	0
	H No. in BCD	T No. in BCD	S No. in BCD	M No. in BCD			2nd spindle override switch available for spindle 2	1st spindle override switch active for spindles 1+2

PLC MD 2001, bits 4 to 7: ”0”= Fixed-point number
”1”= BCD number

Using bits 4 - 7 it may be distinguished whether the values of the corresponding data groups (M No., S No., etc.) are to be transmitted from the NC to the PLC as fixed-point or BCD numbers. (See Interface Description 1/Section on ”output of auxiliary functions NC PLC” for precise value ranges.)

M No. -	No. of M function	(M-word 1-3;	FY 53-64)
S No. -	No. of S function	(S-word;	FY 65-70)
T No. -	No. of T function	(T-word;	FY 71-76)
H No. -	No. of H function	(H-word;	FY 77-82)

Transmission as a fixed-point number is only practical if the value range is not adequate for the data group to be transmitted as a BCD number.

Bit 0 1st spindle override switch active for spindles 1+2

Bit = 0 The PLC operating system does not transfer the signals for the 1st spindle override switch to the interface for the 2nd spindle.

Bit = 1 The PLC operating system transfers the signals of the 1st spindle override switch and the key signals spindle ON/OFF in addition to the interface for the 2nd spindle (QB 104 bits 0..3 and 7).
Transfer takes place before the PLC user program is called. This gives the user the opportunity of modifying the signals in his PLC program. The signals are transferred to the NC only at the end of the OB1.

Bit 1 2nd spindle override switch available for spindle 2

In order to be able to modify both spindle speeds independently of each other, it is necessary to install a 2nd spindle override switch with the associated spindle enable keys.

Bit = 0 No 2nd spindle override switch can be installed.

Bit = 1 A 2nd spindle override switch and keys for spindle ON/OFF are installed. Switch and key signals must be provided at input byte 4. The PLC operating system transfers the signals to the interface for the 2nd spindle.

Spindle override switch	IB 4 bit 0..3 to QB 104 bit 0..3
Taster	IB 4 bit 6+7 to QB 104 bit 7

Bits for PLC MD 2002

Bit 0 No PLC alarm processing

Bit = 0 The system program calls OB2 in the event of edge change in the interrupt input byte. If the OB2 is not available, the PLC alarm 6105 (missing MC-5 block) is triggered.

Bit = 1 The system program ignores the interrupt input byte, OB2 is not called no alarm message.

Bit 1 2nd axis selector switch available

With the SINUMERIK 810M/820 M a 2nd axis selector switch with the necessary traversing keys must be incorporated in the machine control panel in order to be able to traverse 2 axes simultaneously manually. The PLC operating system can then decode the 2nd axis selector switch and traversing keys accordingly. The otherwise freely assignable IB 4, in this case, is occupied by the 2nd axis selector switch.

Bit = 0 2nd axis selector switch not available, IB 4 is free

Bit = 1 2nd axis selector switch available
(advisable only for M version with external machine control panel)

Bit 3 **Transfer machine control panel from input to output process image**

Bit = 0 The machine control panel must be transferred to the output image by the PLC user program.

Bit = 1 The PLC operating system (OS) transfers the machine control panel from the input to the output process image. This transfer takes place before the PLC user program is called. Consequently, the user can still modify the machine control panel signals in his program. The machine control panel signals are not transferred from the PLC to the NC until the PLC program has been completed, i.e. at the end of OB1.

Transfer of the external customer machine control panel by means of the operating system is only advisable if the terminal assignment is identical with that of the Siemens machine control panel.

Corresponding MDs:
- NC MD 5008 bit 0 and bit 1
  Internal or external machine control panel available
- PLC MD 2002 bit 1
  2nd axis selector switch available

Bit 4 **Connection of decentralized I/O device**

Unlike the centralized I/O device, the decentralized I/O device is connected to the 6FX1 138-5BA01 CPU module at connector X111.

The decentralized I/O device has the following address area:

Inputs	IB	0 ... 63
Outputs	QB	0 ... 31
I/O device	PB	127 ... 159 (mini EU only for analog input modules)

In the event of mixed operation involving centralized and decentralized I/O devices, make sure that the centralized I/O device is always addressed before the decentralized I/O device. The interrupt input byte may also be decentralized (distributed).

In terms of hardware, two types of centralized I/O device are available to the user:
Mini expansion unit (mini EU)
Maxi expansion unit (maxi EU)
CU/MPC interface module 6FX1 132-1BA..

Bit = 0 No decentralized I/O device available
Bit = 1 Decentralized I/O device connected

Bit 5	**Rapid traverse/feedrate override for 3rd to 7th axis for T version**

Bit = 0 Axes 3 to 7 always traverse at 100%. No rapid traverse/feedrate override possible.
Bit = 1 Rapid traverse/feedrate override also active for axes 3 to 7 (auxiliary axes).

Bit 7	**M-decoding with extended address**

Bit = 0 The extended M functions are only output on FY 54 to 64. No flag bits are available.
Bit = 1 The extended M functions are also output as flag bits (FY 92 - 99). For this purpose, a decoding list must be written in DB 80. This list specifies the assignments between the M functions and flag bits. Up to 32 extended M functions can be defined in DB 80.

Bits for PLC MD 2003

Bit 0	**PLC STOP with time-out in OB2**

Bit = 0 If the interpreter run time exceeds the value specified in PLC MD 3, alarm 6160 (time-out OB2) is triggered.

Bit = 1 Interpreter run time > PLC MD 3
- PLC goes into STOP condition
- Alarm 6160 displayed

Bit 1	PLC STOP with time-out OB1 + OB2

Bit = 0 Interpreter run time > PLC MD 1
- Only alarm 6159 (time-out S 5 program)
- PLC cycle time may exceed 60 ms

Bit = 1 Interpreter run time > PLC MD 1
- PLC moves to STOP
- Alarm 6159 displayed

Bit 2	PLC STOP if decentralized I/O device faulted

This bit allows the user to decide whether the PLC assumes the Stop state when the decentralized I/O device exhibits one of the following faults:

- Expansion unit fails to respond (Alarm 6138)
- Data transmission to decentralized I/O device faulted (Alarm 6139)
- Change in decentralized I/O device (module failure 6147)

Bit = 0 In the event of one of the above faults only the appropriate alarm is displayed.

Bit = 1 In the event of one of the above faults the PLC assumes the STOP state and the appropriate alarm is also displayed.

Overtemperature in the expansion unit does not give rise to PLC STOP, irrespective of MD 2003 bit 2.
Only alarm 6148 (overtemperature in EU) is displayed.
The user can then respond via group alarm M 8.0.

Bit 3	Warm restart with PLC-STOP

Bit 3 = 0 The control activates ”Cold restart” on return of voltage.
Bit 3 = 1 On return of voltage, the PLC continues the program at the interrupt point after mains failure.

SINUMERIK 810/820 T/M startup behaviour

Introduction

The interfaces between the SINUMERIK 810/820 T/M operating system and the PLC 135W program are the organisation blocks. These blocks, being part of the PLC programming language PLC 135W, can only be called by the operating system of the control. The user can program the relevant organisation blocks and thus indirectly influence the control process between the control and the machine.

For the SINUMERIK 810/820 three organisation blocks are available:

a) OB 1 – Cyclic processing of user program.
b) OB 2 – Interrupt-driven processing of user program.
c) OB 20 – Automatic cold restart in the event of NC overall restart or automatic warm restart on return of voltage.

Using OB2 the user can indirectly influence the startup behaviour of the SINUMERIK 810/820 T/M. There are 3 possibilities: overall reset with cold restart, automatic cold restart and warm restart.

Overall reset

In the startup mode ”Overall reset” all RAM areas for PLC are cleared. This ensures a completely new start of the user program. FBs or DBs must be reloaded, for example.

Cold restart is one of the components of the PLC overall reset mode. However, the organisation block OB20 is not processed in this startup mode. When the ”End of installation” key is operated, an automatic cold restart takes place.

Automatic cold restart

The user can activate automatic cold restart with NC overall reset. With overall reset and automatic cold restart the SINUMERIK 810/820 T/M operating system performs the following actions:

- Clearing of all current time values
- Clearing of all current counts
- Resetting of all flags and presetting of interface flags
- The address lists of the relevant blocks required for the PLC 135W programming language (e. g. data blocks) are drawn up anew.

The minimum set of data blocks required

- DB 1 ”System measuring data”
- DB 36 ”NC data channel”
- DB 37 ”Triggering of the two V.24 (RS232C) interfaces” are reached.
- DB 39 Software cams
- DB 40 Display functions
- Sum check of STEP5 blocks
- Clear and reinitialize I/O device configuration
- Clear memory areas provided for function blocks FB 61 and FB 62 (NC/COM PLC):
- Load machine data and check syntax
- Calculate the STEP5 auxiliary times on the basis of the position control clock pulse.
- Clear and preset VDI signals for NC/spindle.

The operating system calls automatic cold restart with OB20 on actuation of the ”Startup end” key. The PLC operating system then calls the user program OB1-Start. With the PLC functions ”STOP” and following ”START”of the PG, automatic cold restart is performed.

Warm restart

Warm restart (MD 2003 bit3=1) is automatically started by the system program of the SINUMERIK 810/820 T/M on mains failure and subsequent ”Power on”. The operating system first calls organisation block OB20. After OB20 has been processed, the cyclic program is continued with valid inputs at the interrupt point. Flags, timers and counters remain valid.

Zero signals are then issued for all outputs at the end of the specified cycle. The system program then calls OB1.

On warm restart the following operations are performed:

- Load saved block pointers and interrupt stack pointers into the relevant arithmetic coprocessor registers.
- Clear process image of inputs
- Clear and reinitialize basic flags FY0 ... FY24 (zero for cold restart)
- Clear reentrant data areas, e. g. for FB 61 and FB 62
- Clear reentrant data area PLC OS
- Clear NC PC buffer memory
- Load arithmetic coprocessor with the specified number of STEP5 timer locations.
- Clear and preset VDI signal
- Start decentralized I/O device
- Draw up I/O device configuration list of the I/O submodules
- Summation of all blocks
- Call OB20 (outputs are not issued), then load arithmetic coprocessor with the pointers of the user programs's interrupt point.

On warm restart, the PLC machine data have the same validity as before mains failure. The PLC machine data are updated only by cold restart.

Bits for PLC MD 2003

Bit 4 Enabling of S5 system commands

System commands (in FBs only) allow direct accessing of specific addresses in the control's memory area. System commands have been bit-disabled on account of the differences in usage compared with an S 5 control.

Bit = 0 LIR; TIR; TNB; TNW disabled
Bit = 1 LIR; TIR; TNB; TNW enabled

Correct use of the system commands is described

in the Programming Guide.

Bit 5 Demonstration mode

If this bit is set, any available user program (OB1; OB2) is no longer processed. The operating system then calls a demo program held in the assembler to supply the following signals with a ”1” signal:

- Emergency Stop
- Spindle override active
- Servo enable Axes 1 – 7
- Feed enable Axes 1 – 7
- Feedrate override active
- Read-in enable Channels 1+2
- Axis disable Axes 1 – 7

Demo mode is intended for demonstration purposes only!

Interrupt processing MD 2002 bit 0 disabled?

Bit = 0 No demo mode, PLC operating normally.
Bit = 1 Demo mode selected. OB1 + OB2 no longer processed.

Bits for PLC MD 2003

Bit 6 Segmentation of S 5 program processing

This MD bit is used to match the PLC user program to the machine requirements.

Bit = 0 No segmentation
PLC cycle time = 60 ms (12 x sampling time of position controller),
i.e. interrupt programs and cyclic programs must not exceed the maximum permissible interpreter run time of 9.0 ms (with MD 1 = 15 %) within 60 ms.

Bit = 1 Segmentation
The maximum permissible interpreter run time in PLC MD 1 is always maintained relative to a 20 ms interval (4 x sampling time of position controller) (i.e. a segment of the PLC program is processed every 20 ms until completed). The cycle time may exceed 60 ms.

Example: Calculation of cycle time given a PLC interpreter run time of 12 ms
MD 1 = 15 %, MD 2003 bit 6 = 1
15 % of 22 ms = 3.3 ms (e.g. IPO = 22 ms if NC MD 155 = 3)

$$\frac{12.0\text{ ms}}{3.3\text{ ms}} = 3.64 \qquad \text{4 segments}$$

Cycle time: 4 segments x 22 ms = 88 ms

Bits for PLC MD 2003

Bit 7 — Enabling of diagnostics function (stored in DB1)

This MD bit enables the diagnostics function for run-time measurements of the PLC program.

Bit = 0 Diagnostics function disabled
(Note: Reset bit to zero after installation)

Bit = 1 Diagnostics function enabled
(Note: Enable only during installation)

The diagnostics DB has its own initiation bit; enabling of the diagnostics function alone is not sufficient.

Description of diagnostics DBs

DW 0:	Current cycle time			[ms]
DW 1 :	Current interpreter run time OB1 + OB2			[µs]
DW 2 :	Current cyclic interpreter run time OB1			[µs]
DW 3 :	Current interrupt-driven interpreter run time OB2			[µs]
DW 4 :	Number of interrupt processing operations per cycle			
DW 8 :	Bit	0	EU does not respond	
	Bit	1	Decentralized I/O device transmission faulted	
	Bit	2	Overtemperature in EU	
	Bit	3	Change in decentralized I/O device	
	Bit	8	Time-out OB1 + OB2	
	Bit	9	Time-out OB2	
	Bit	11	Failure of master PLC	
DW 9 :	Bit	15	Initiation bit for diagnostics	

The measured values refer to the present cycle time in DW 0 (cycle time   60 ms).

for DW 8 : If the PLC does not assume the Stop state in the event of the alarms in DW 8, group bit F 8.0 is set in the basic signal flag area. (The S 5 user program then activates the diagnostics DB, reads out DW 8 and responds accordingly.)

Initiation possibilities for diagnostics function

DB 1
DW 0 KF = Cycle time (ms)
DW 1 KF = Interpreter run time OB1 + OB2 (µs) *
DW 2 KF = Cyclic interpreter run time OB1 (µs) *
DW 3 KF = Interrupt-driven interpreter run time OB2 (µs)

- Periodical display via PG function
  STATUS VAR and corresponding initiation program in OB1

```
OB 1
C DB 1       Call diagnostics DB
A F 0.1      "1" flag
= D 9.15     Initiation bit
  .
  .
  .
  .
BE
```

DB 1 is called and updated cyclically. In this way the variations in the run times can be readily assessed (cycle time and interpreter run time are not constants!).

Only set bit 7 to "1" during the installation phase.

After installation it should be reset to "0"!

PLC MD 3000 ... 3003	User MD bytes

These 4 bytes are freely assignable by the user. With each COLD RESTART of the PLC they are transferred to flag bytes 116-119 where they may be processed by the PLC program.

PLC MD		Flag byte
3000	→	FY 116
3001	→	FY 117
3002	→	FY 118
3003	→	FY 119

---

* *KF only up to 32767, represent higher values in KH!*

6.3 PLC installation

6.3.1 General

In overall reset SET UP MODE the PLC is in the Stop state as the default value. It can be started via the programmer. The PLC is then capable of operating but with the following limitations:

- PLC NC interface is not transmitted to NC
- NC PLC interface is not transmitted to PLC

The NC/PLC interface is rendered inactive for the NC and PLC.

With the PLC in the Stop state the outputs may be addressed via the programmer function CONTROL. The PLC overall reset function (softkey) only operates when the PLC is in the Stop state (as with overall reset with the programmer).

See Section 5.1 for the standard installation sequence.

6.3.2 NC machine data for PLC

The following NC MD and MD bits are of particular significance for the PLC and may have to be modified with respect to the standard (default) values.

1) NC MD 155 Position controller sampling time

NC MD 155	PLC cycle time without segmentation	Interrupt control
0	48 ms	4.0 ms
1	54 ms	4.5 ms
2	60 ms	5.0 ms (standard value)
3	66 ms	5.5 ms
4	72 ms	6.0 ms
5	78 ms	6.5 ms

2) NC MD 5008

Bit	7	6	5	4	3	2	1	0
NC MD 5008					External customer m.c. panel	Internal handwh. selection	External machine ctrl. panel	Internal machine ctrl. panel

Bit 0 Internal SIEMENS machine control panel or internal operator panel attached
Bit 1 External SIEMENS machine control panel attached
Bit 2 Internal handwheel selection using softkeys
Bit 3 External customer machine control panel attached (not an original SIEMENS panel) Difference in decoding of the rotary switches in the two machine control panels. PLC MD 2002 bit 3 must not be set.

3) NC MD 540* Channel-specific bits (*= channel number)
540* bit 0 Auxiliary functions to PLC

4) NC MD 546* Channel-specific bits (*= channel number)
546* bits 0-7 These MD bits control auxiliary function output from the NC to the PLC.

5) NC MD 5004 bit 4 Separate rapid traverse override available. If this bit is set, QB 85 must be taken into account.

6) NC MD 5012 bit 7 PLC alarm texts 6000 - 6063 and 7000 - 7063 input (tape or programmer).

For PLC MD and PLC MD bits see Section 6.2

6.3.3 Link PLC PG 750/685/675/670/635/615

PG 750/685/675/670/635/615 may only be operated at the 1st interface (X121 on CPU module).

1. Press ”SETTING DATA” softkey
2. Key $\boxed{>}$ Extend softkey menu
3. Press ”SE-BITS” softkey

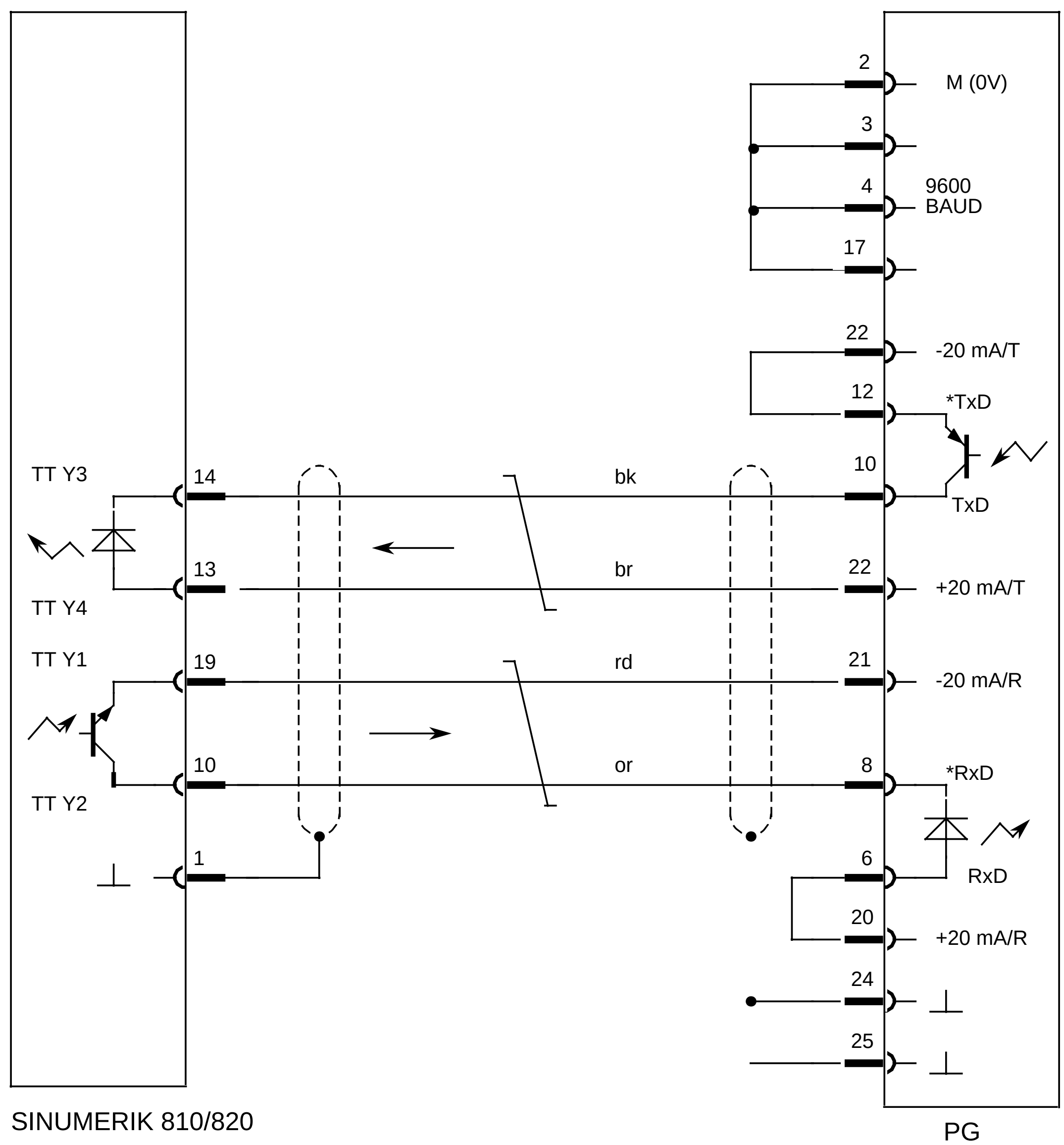
   SD 5010 ... 00000100 Device identifier
   SD 5011 ... xx xxx 111 9600 baud (x ... no significance)

   In this way the 1st interface is interfaced to the PG.
4. Key $\boxed{\wedge}$ Recall. Return to basic display
5. Press ” DATA IN-OUT” softkey

   Select 1st interface (input ”1”).
   Interface assignment of 1st interface must contain ”PLC-PROG”.
6. Press ” DATA IN START” softkey
   This activates the 1st interface in the NC.
   There is no checkback signal on the NC screen unless the ”DATA IN START” softkey is pressed twice. When pressed twice, the input message ”Interface busy” is displayed. Alarm 22 (V.24 time monitor) is rendered inactive in this mode, in addition to the interface signal ”V.24 running”.
7. Key $\boxed{\wedge}$ Recall. Return to basic display

Connect PG to the 1st interface using the cable described and select S5-150S (or language subset B) in the presettings.

Cable name: **PG 685/675/670/635/615 PLC programmer, PLC 55 interface 1st NC X121 interface**
Order No.: **6FC9 340-8G**

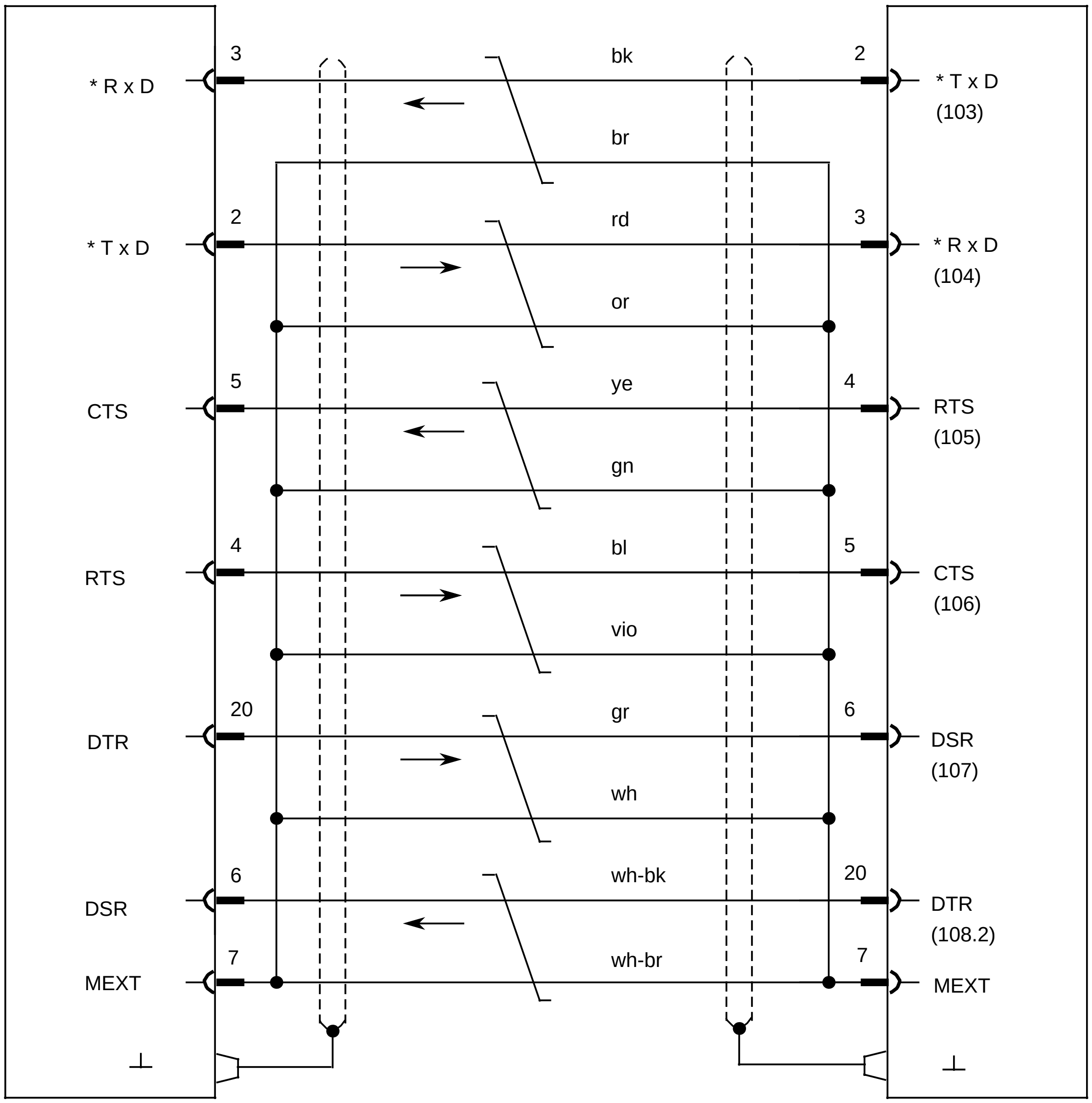


The PLC and PG are now interfaced. Items 1-4 are not applicable when the PLC is linked once more with the PG following Power On.

A special module such as the AS 511 (interface module) is not required in the SINUMERIK 810 GA3/820 GA3 as all functions required for this module are already available in the control.

The cable can be used for the PG 750 PLC link if the adaptor supplied with the PG 750 is used.

Cable name: SIMATIC PG 730/750 (TRANS PCIN and PLC programming)
PC (AT compatible) 1st interface NC x 121
Order No. : **6FC9 344-4R**



SINUMERIK 810/820 PG connector COM 1

6.3.4 Programmer commands

The following functions are available in the control:

- Functions operating only in PLC STOP:
  - INTERRUPT STACK (I stack) output
  - BLOCK STACK (B stack) output
  - Overall reset (CLEAR PLC, block B)
  - PLC cold restart (PLC START) (not in the OVERALL RESET mode)
  - CONTROL (outputs, e.g. QB0; only the outputs are controlled, not the process image)
- Functions also operating in cyclic operation:
  - Block input
  - Block output
  - STATUS VAR
  - STATUS block
  - CONTROL VAR
  - SYSPAR (see Programming Guide)
  - PLC ENTRY inquiry, blocks OB, PB, FB, SB, DB
  - Output cross-reference list
  - SPAUS PLC (see Programming Guide)
  - PLC STOP
  - Delete blocks (CLEAR PLC)
  - Compress memory (COMP PLC)

The following functions are not available:

- Program check
- PLC warm restart (general)

6.4 PLC operating system (OS)

6.4.1 The following functions are available in PLC OS:

- Processing of error messages (Nos. 6000 - 6063)
- Processing of operational messages (Nos. 7000 - 7063)
- M-decoding into static and dynamic flags
- Transfer of integrated machine control panel to PLC input image
- Decoding of travel instructions and transfer to IB 99
- External data transfer PLC NC with standard FBs
- Transfer of machine control panel from input image to output image with PLC MD bit 2002 bit 3 set.
- PLC STATUS on NC screen
- Transfer between NC/PLC interface and PLC user interface
- Handwheel selection using softkeys
- After PLC overall reset the following blocks are created by the operating system:

DB 1	Diagnostics DB	FB 11	Create DB
DB 36	Data transfer	FB 40 ... 45 [1])	Tool management (no function macros)
DB 37	RS232 auxiliary DB	FB 60	Block transfer
DB 39		FB 61	Read NC data
DB 40		FB 62	Write NC data
		FB 66	Save flags
		FB 100 ... 104 [1])	Integrated auxiliary (no function macros)
		FB 190	Host PLC

Note: [1]) *Option*

6.4.2 Test mode - demo program

If PLC MD 2003 bit 5 is set to 1, the PLC user program (OB1 + OB2) is no longer processed by the PLC operating system (OS). The PLC operating system then generates the necessary interface signals to permit test operation, even without I/O modules. In this condition NC part programs can be processed, the actual axis value displays being updated.

NC Start acts only on channel 1!

Preconditions:

- NC start without reference point: NC MD 5003 bit 3 = 1
- Set NC MD 2000, 2001, 2002, 2003 to 0
- Disable interrupt processing OB2, PLC MD 2002 bit 0 = 1

Note:

Test mode with the demo program is intended only for demonstration purposes without a machine.

6.5 PLC STATUS

For servicing and test purposes the PLC operating system incorporates a PLC status display with which the condition of all inputs, outputs, flags, timers and counters can be indicated and altered on the NC screen.

	Read	Write	Data No.
Inputs	X	X	0 ... 126
Outputs	X	X	0 ... 126
Flags	X	X	0 ... 254
Timers	X	no	0 ... 31
Counters	X	no	0 ... 31
Data words	X	X	0 ... 255

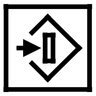
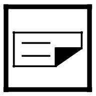
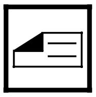
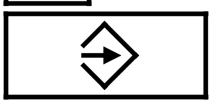
Writing of timers and counters with PLC STATUS is not possible.

Writing of inputs, outputs and flags is only possible once the password has been entered.

Selection of PLC STATUS

”DIAGNOSIS” softkey
”PLC STATUS” softkey
Enter password ”...........” if necessary

Operation of PLC STATUS for SINUMERIK 810 GA3

Key:		PRESELECTION:	preselection possible to any existing byte number
Key:		UP:	Page number incremented by one
Key:		DOWN:	Page number decremented by one
Key:	→	RIGHT:	Move cursor to next field
Key:	←	LEFT:	Move cursor to previous field
Key:	↓	DOWN:	Move cursor to next lower field
Key:	↑	UP:	Move cursor to next upper field
Key:		INPUT:	Change value in selected word or bit number
Key:	∧	RECALL:	Return to previous display
Key:	>	EXTEND SOFTKEY MENU	
Key:	0 / S	Number 0	
. . .		. . .	
Key:	9 / Z	Number 9	
Key:	%	Number a	
Key:	/	Number b	
Key:	N	Number c	
Key:	G	Number d	

Operation of PLC STATUS for SINUMERIK 810 GA3 continued

Key:	Q	or	4	Number e
Key:	B	or	U	Number f

Operation of PLC STATUS for SINUMERIK 820 GA3

Key:	0	...	9	Numbers 0 to 9
Key:	—	+	A	Number a
Key:	—	+	B	Number b
Key:	—	+	C	Number c
Key:	—	+	D	Number d
Key:	—	+	E	Number e
Key:	—	+	F	Number f

Hexadecimal inputs in KH format must be entered in lower case (as for SINUMERIK 810).

Softkey:
- IW = Input words
- QW = Output words
- FW = Flag words
- T = Timers
- C = Counters
- DB = Data blocks
- DW = Data words
- KH = Hexadecimal representation
- KM = Bit pattern representation
- KF = Fixed-point number
- KS = ASCII characters

PLC STATUS display

Word condition code (e.g. output word 10 and flag word 90) :

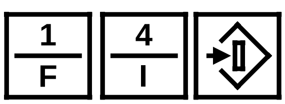
Word	QW10		FW90	
Byte	QB10	QB11	FY90	FY91
Bit	7 6 5 4 3 2 1 0	7 6 5 4 3 2 1 0	7 6 5 4 3 2 1 0	7 6 5 4 3 2 1 0
KM	0 1 1 0 0 0 1 0	0 0 1 0 0 1 0 1	0 0 1 1 0 0 0 1	0 0 1 1 1 1 1 1
KH	62	25	31	3f
KF	25125		12607	
KS(for DW only)	b	%	1	?

Reading of input, output or flag words

Softkey: IW, QW, or FW

Example for: SINUMERIK 810 GA3

Keys:



Preselection to word No. 14

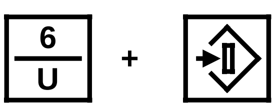
Softkey: KH or KM to switch display format to hex or binary

Reading of timers and counters

Softkey: T or C

Example for: SINUMERIK 810 GA3

Keys:



Preselection to timer or counter No. 6

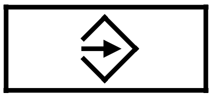
The time is displayed in STEP 5 representation.
The count is indicated in BCD code.

Writing of input, output, flag and data words

Writing is only possible once the password has been entered.

Softkey: KH, KS or KM to switch the display format if necessary.

Keys: Input of desired value in format:
KM ... Numbers 0 and 1
KH ... Numbers 0 to f (lower-case letters)
KS ... Alphanumeric (for data words only)
Leading zeros need not be entered.

Key:  Input

7 Machine Interface

7.1 Actual-value input measuring circuit

(6FX1 121-4B . . module)

7.1.1 Connector pin assignment

The actual values are supplied to the NC via 15-pin connectors. 6FX 1121-4B . . measuring circuit.

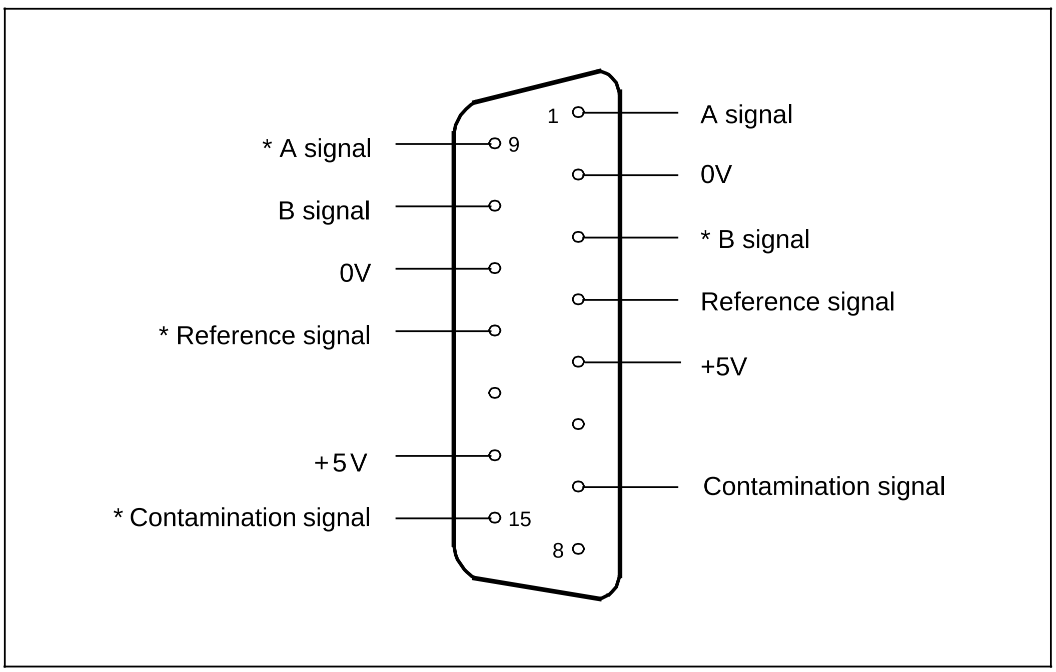
X131	Actual-value connector 1	Encoder 1
X111	Actual-value connector 1	Encoder 2
X121	Actual-value connector 1	Encoder 3

Incremental rotary encoders (e.g. ROD426) for linear axes or incremental linear encoders with EXE external pulse-shaper electronics (e.g. linear scale LS703 and EXE603) are connected.

The input signals to the measuring-circuit modules are identical for both encoders.

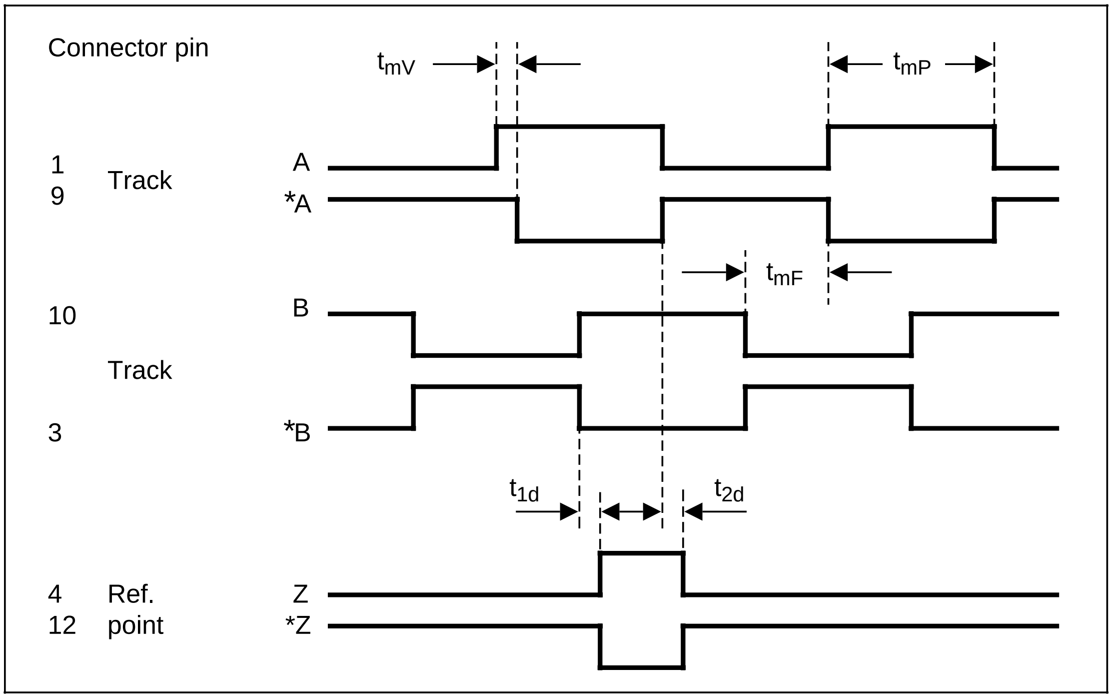
03390 and 03395 modules are also available with integrated EXE.

In this case the signals are routed from the measuring head directly to measuring-circuit modules and are converted at the module into TTL signals.



7.1.2 Differential input

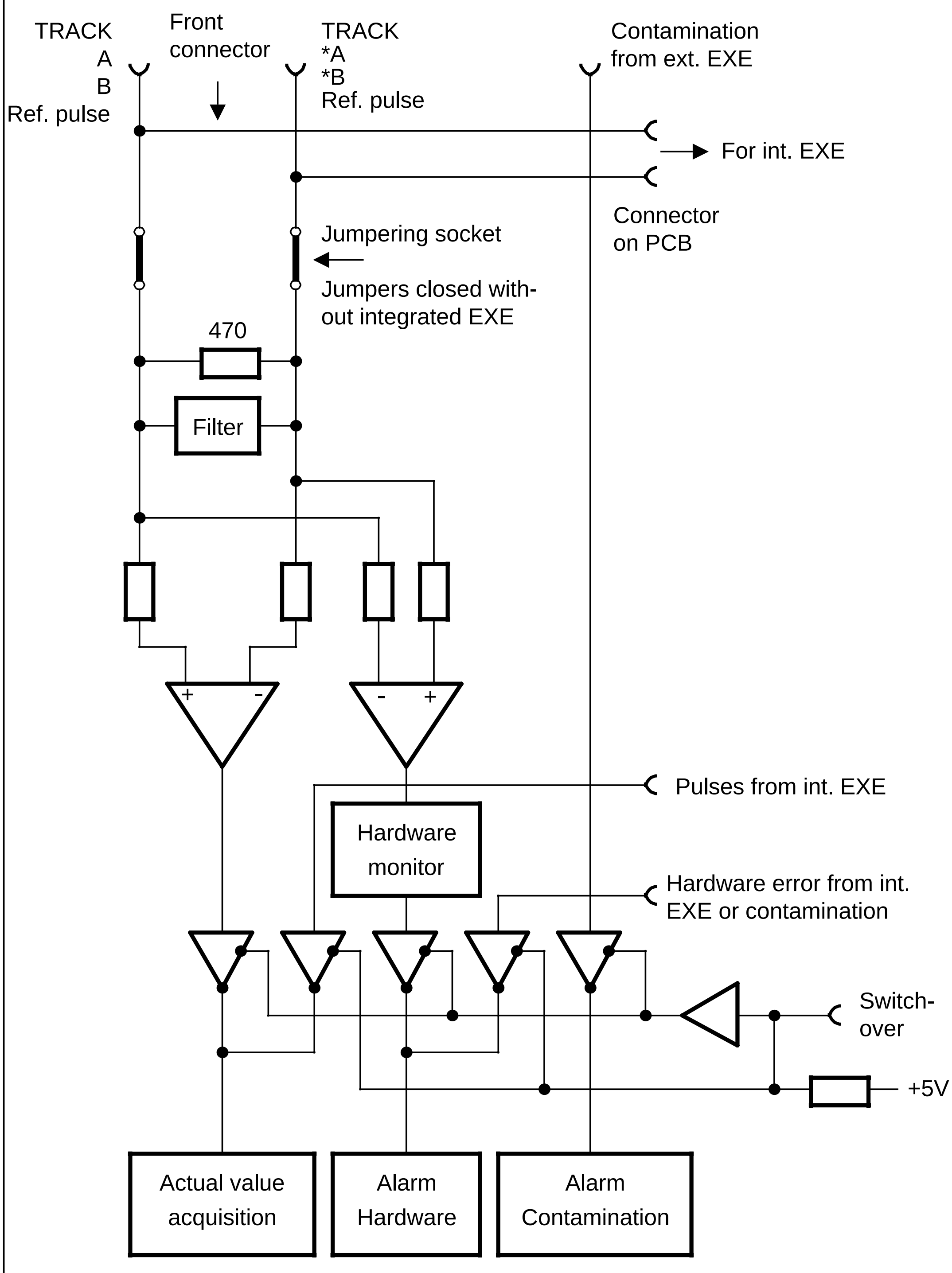
Input signals and characteristics for digital measuring systems with differential output.



Some important data:

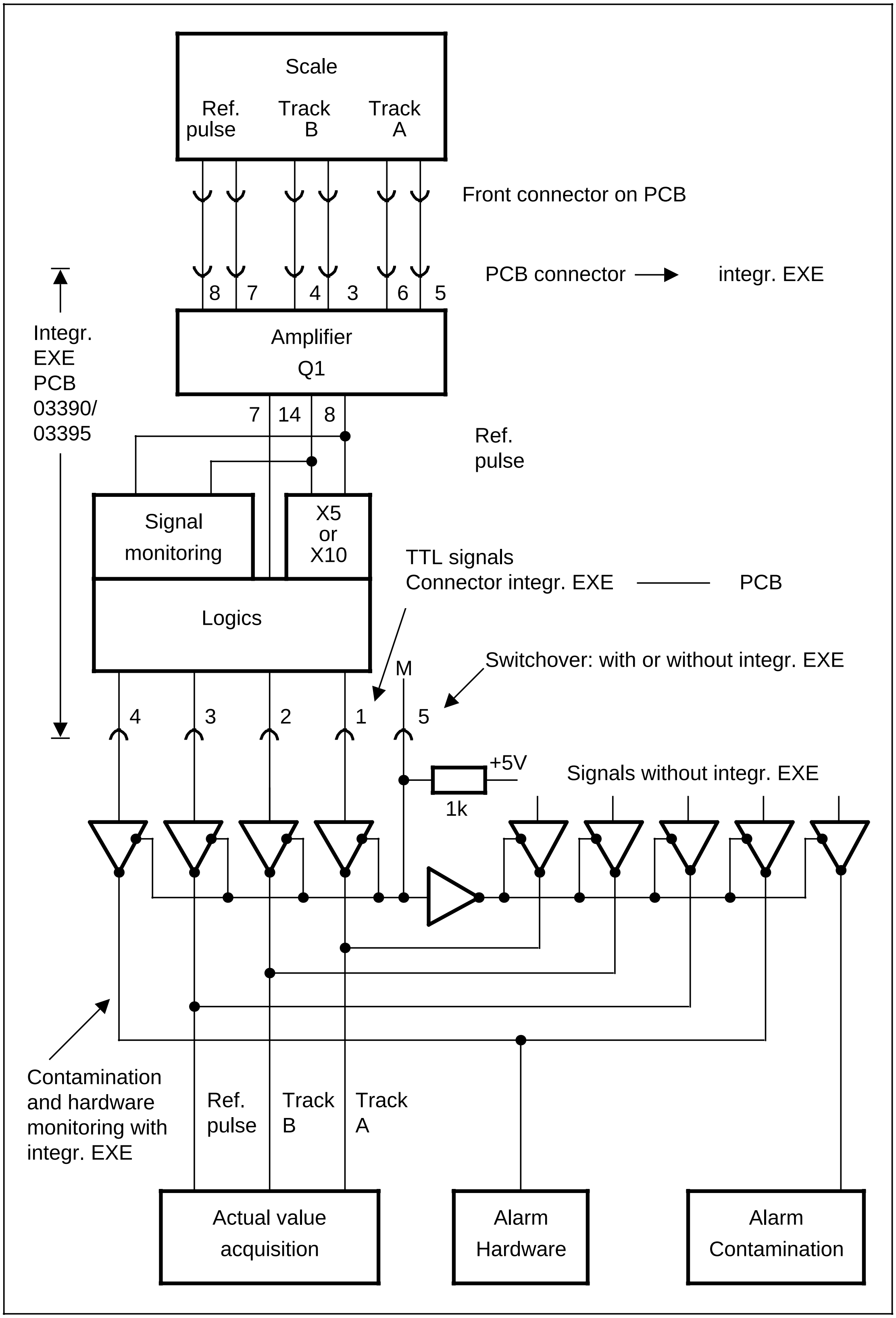
- Maximum cable length to encoder 35 m (for SINUMERIK cables)
- Encoder supply voltage 5V + 5 %
- Current per encoder system 300 mA
  Ohmic input resistance 470 ohm
- Differential input voltage
  e.g. between A and *A 1V
- Differential input voltage max. 10V
  Maximum input frequency at 90° phase displacement between A and B track pulses 1000 kHz
- Minimum pulse width t_{mP} 1 µs
  Minimum interval between two consecutive edges t_{mF} 200 ns
- t_{1d} and t_{2d} 60 ns
- Max. time delay of two consecutive edges of a track t_{mV} 20 ns

7.1.3 Equivalent circuit diagram with differential input



One track and monitor without integrated EXE are shown

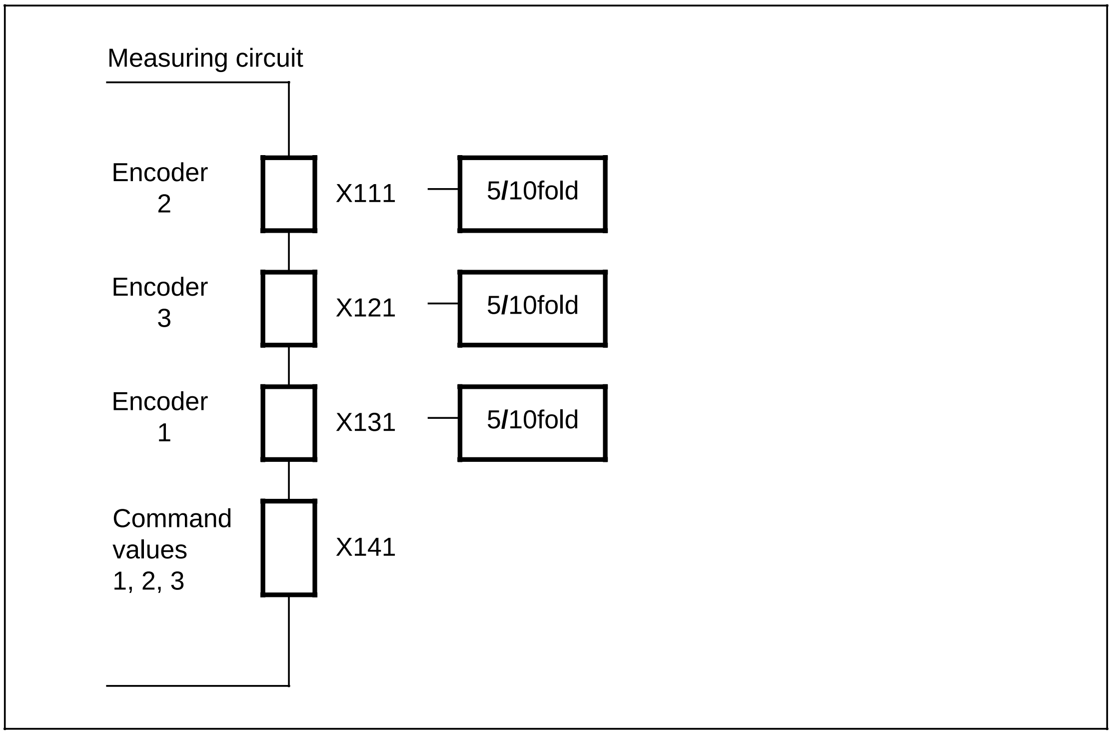
7.1.4 Equivalent circuit diagram for actual-value input with integrated EXE



7.1.5 Complement with integrated EXE

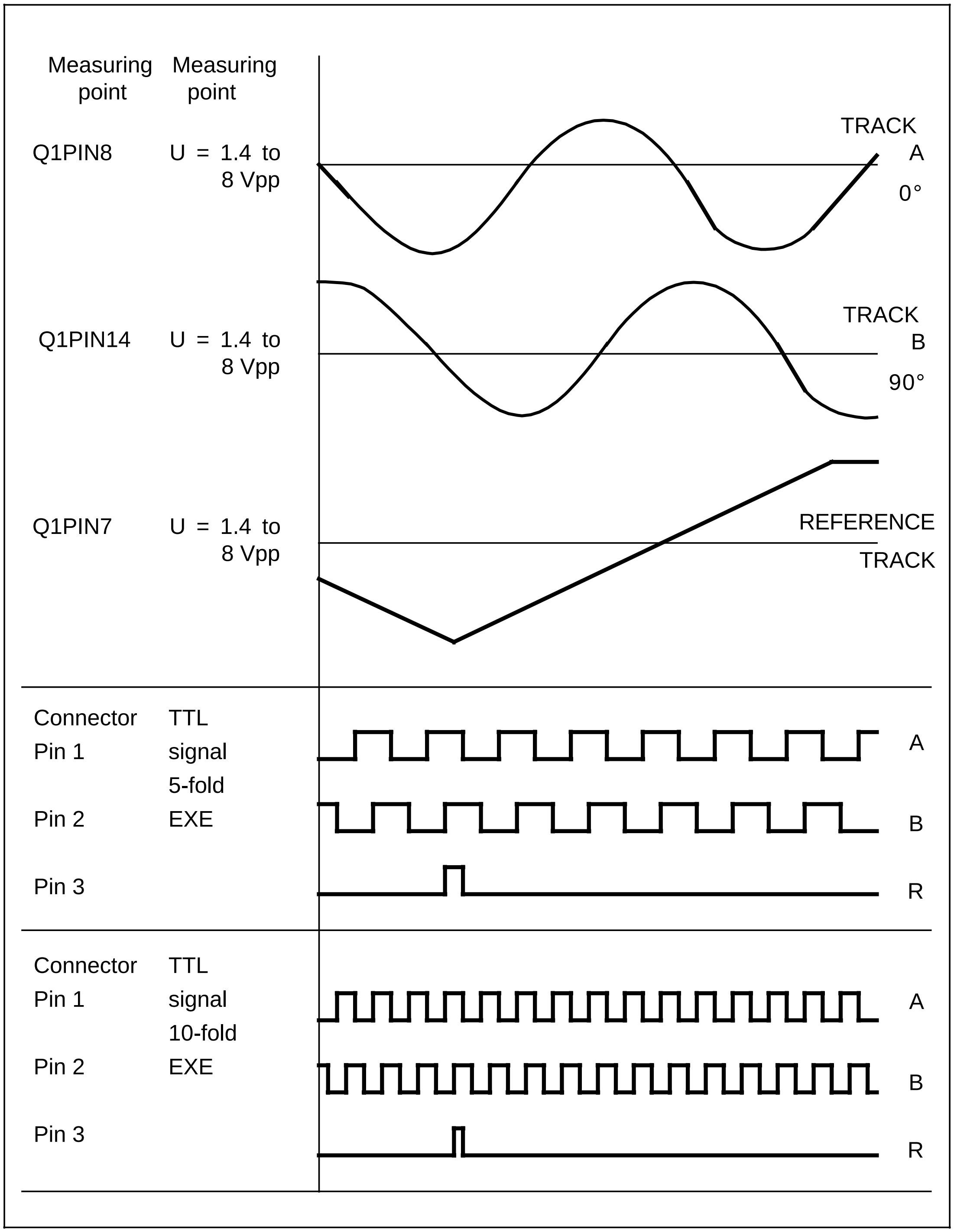
A complement of max. three 5/10-fold EXEs is possible with the 6FX 1121-4B . . measuring circuit.

The permissible combinations with the maximum complement are as follows:



3 narrow slots are reserved for the measuring-circuit modules in the case of the SINUMERIK 810 GA3. Consequently, it is only possible to provide one measuring-circuit module with integrated EXEs since this module then occupies two slots as a result of the 40 mm width.

7.1.6 Input signals with integrated EXE



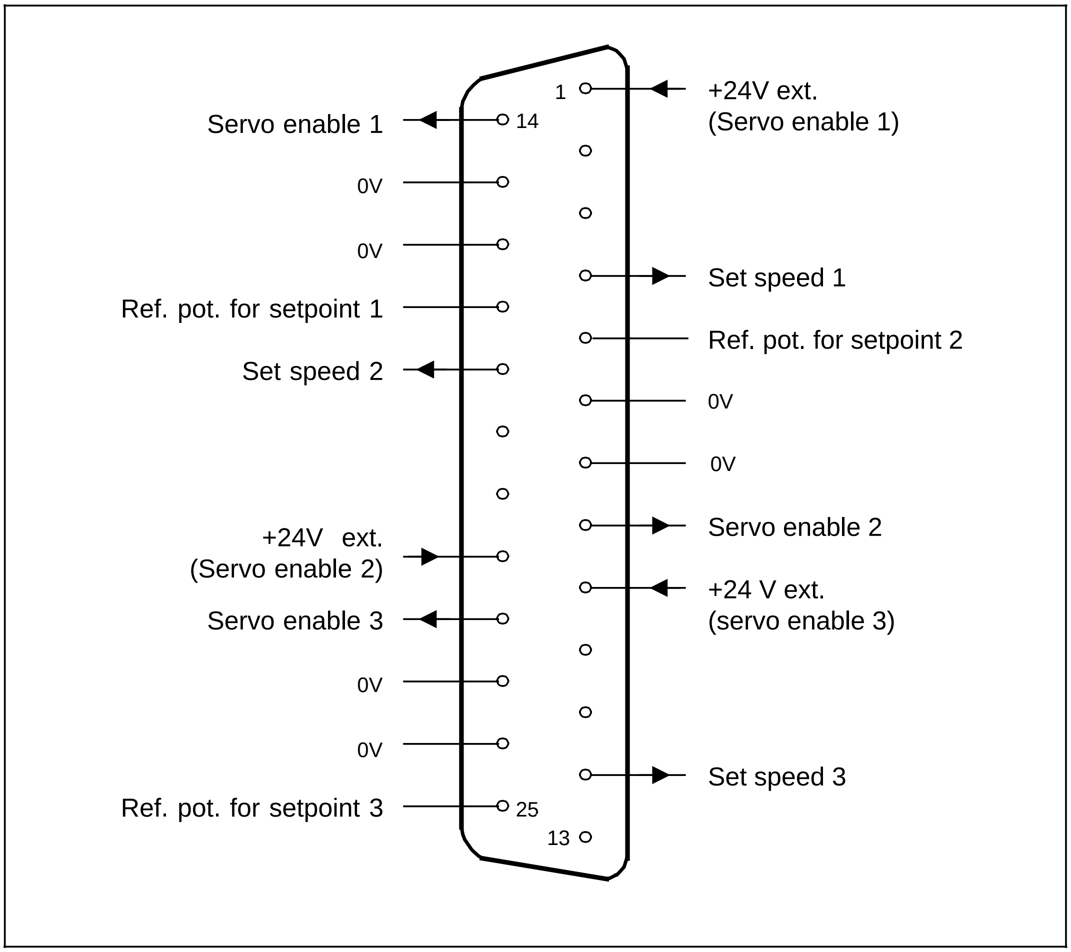
Additional data:

- Measurement of Q1 signals performed earth-free with respect to measuring point U_o
- Phase angle 90 ± 12° from 0° / 90°
- Response threshold for fault signal U = (0.7 ± 0.5) Vpp, behind amplifier Q1.
- Signal size input EXE tracks A and B approx. 11 µA. Ref. track approx. 3.5 µA

7.2 Setpoint output measuring circuit (6FX1 121-4B...)

7.2.1 X141 connector pin assignment

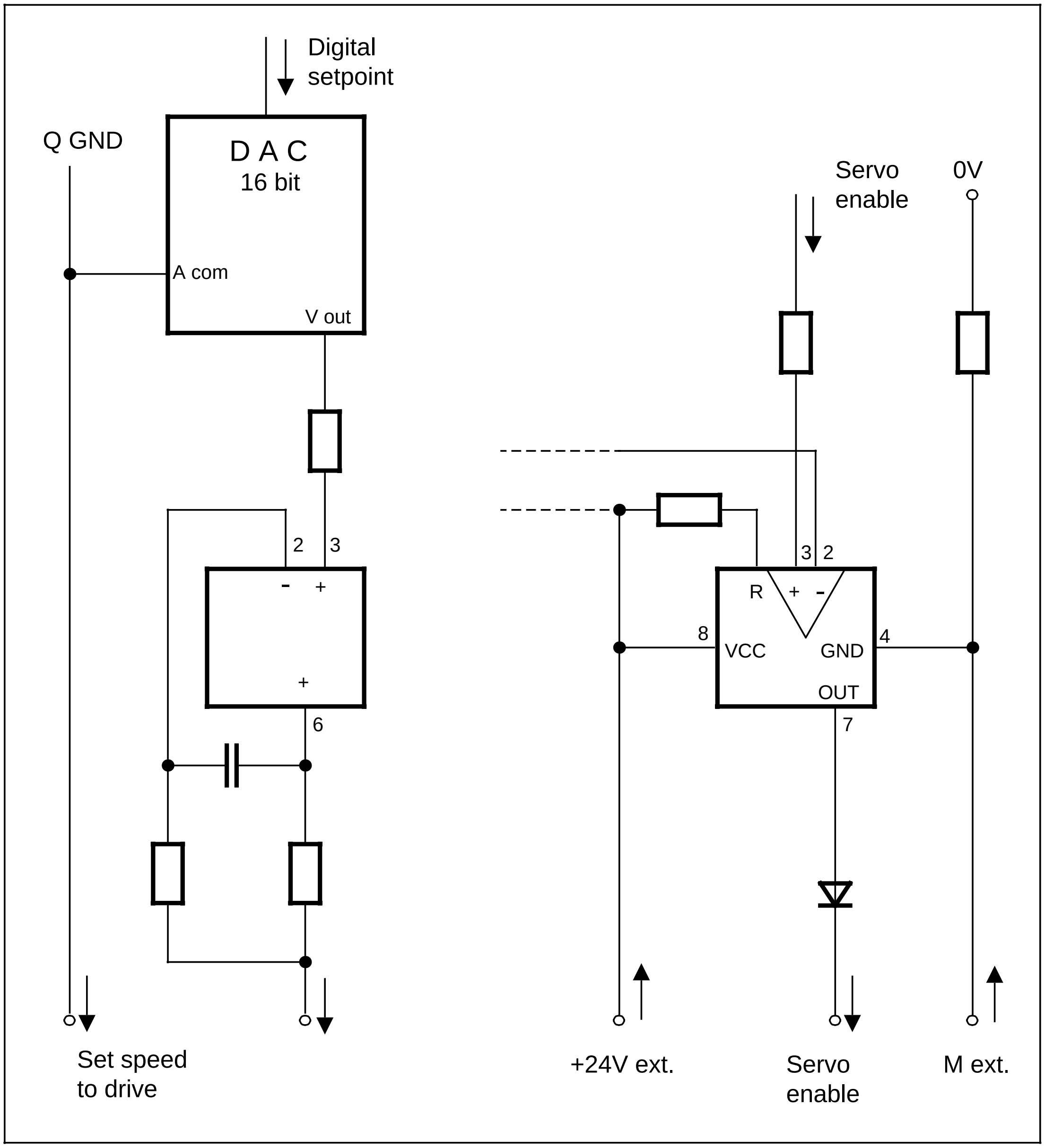
The set speeds and axis-specific servo enable signals (non-floating outputs) are output from the NC via a 25-pin connector.



Characteristics:

- Max. analog setpoint voltage ± 10V
- Max. current 2 mA
- Max. current for servo enable signals (non-floating 20 - 30V) 100 mA (short-circuit-proof)

7.2.2 Equivalent circuit diagram (for one axis)



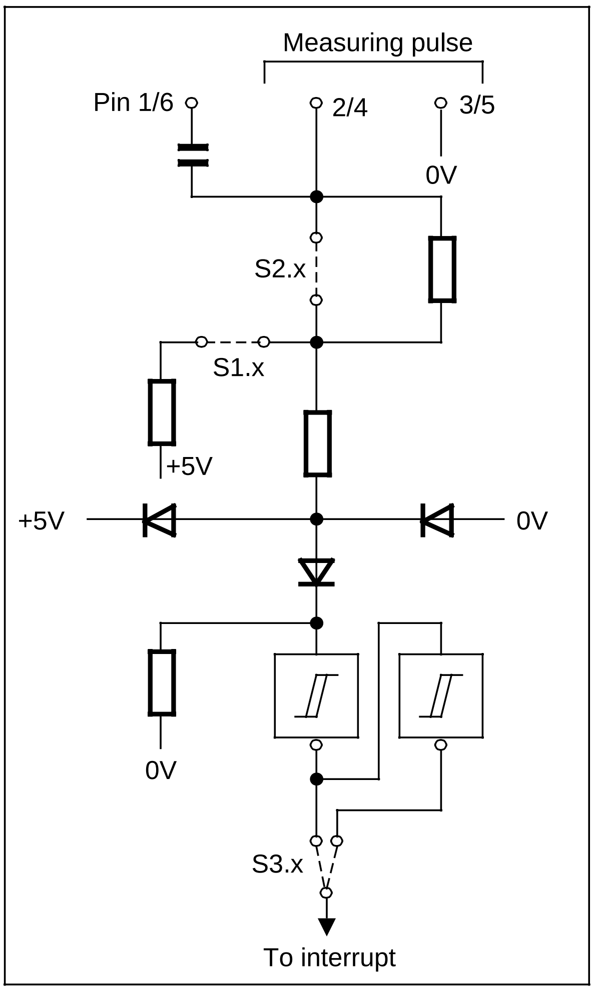
The setpoint output is suitable both for differential inputs and for non-floating inputs. If the drive actuator does not have a differential input, output Q GND should be connected to 0V.

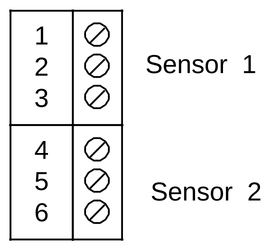
7.3 Sensor input

Only the two sensor inputs on the 6FX 1121-2BC . . interface module may be used at the control (see also Section 3).

The sensor signal triggers an interrupt in the NC, causing the instantaneous actual value to be stored immediately and the axes to be braked after the acceleration ramp.

- **6FX 1121-2BC02 interface module**





S1.x, S2.x, S3.x

Sensor 1 ... S1.1, S2.1, S3.1
Sensor 2 ... S1.2, S2.2, S3.2

Sensor	Pulse edge	S.1.x	S.2.x	S.3.x
24 V	↑	Open	Open	N
	↓	Open	Open	P
TTL	↑	Closed	Closed	N
	↓	Closed	Closed	P
Open collector	↑	Closed	Closed	N
	↓	Closed	Closed	P

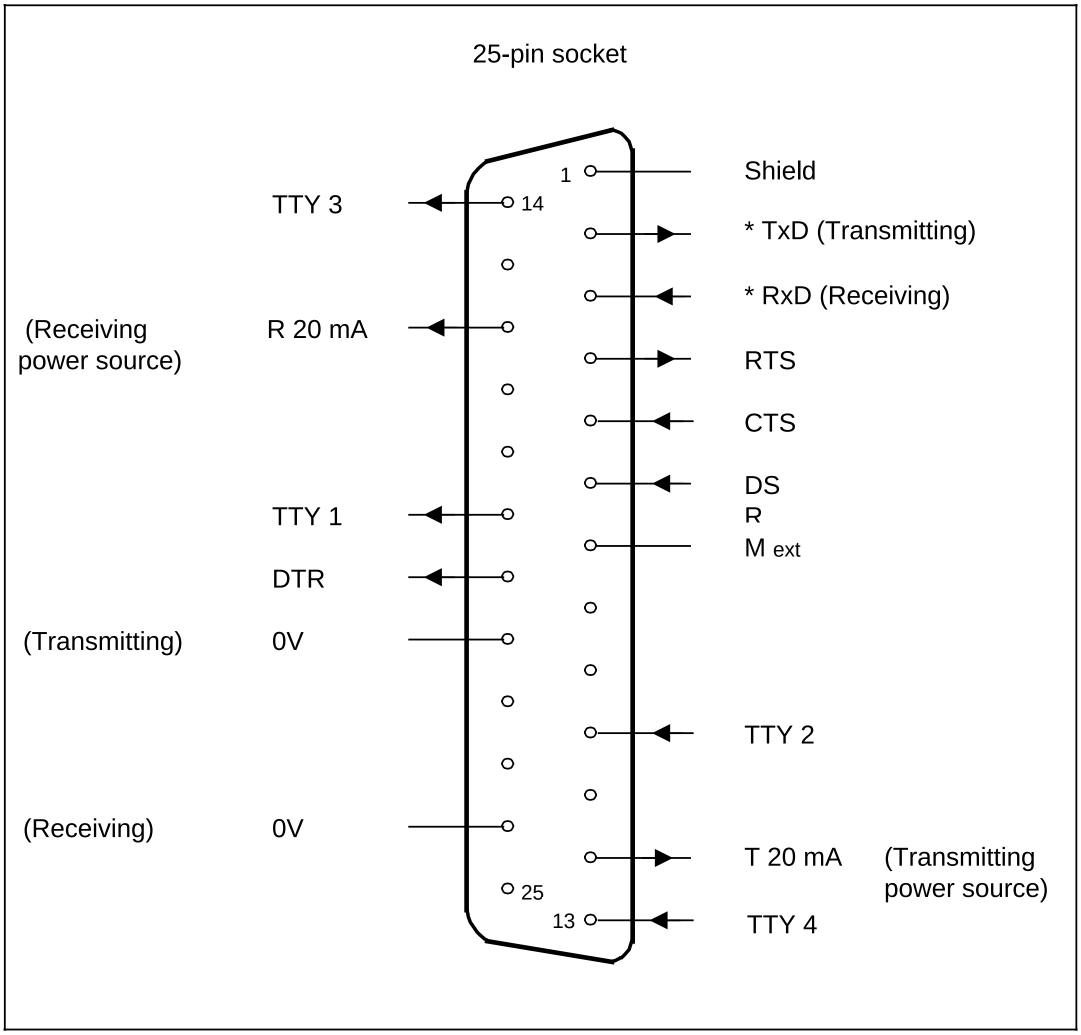
7.4 Serial interface (RS 232/V.24+20 mA)

The System 810 GA3/820 GA3 has 2 serial interfaces:

1st interface (X121): RS232C (V.24) + 20 mA
2nd interface (X131): RS232C (V.24) only

Only full duplex operation of the 20 mA interface is possible.

A detailed description of the serial interfaces is included in the UNIVERSAL INTERFACE Planning Guide.

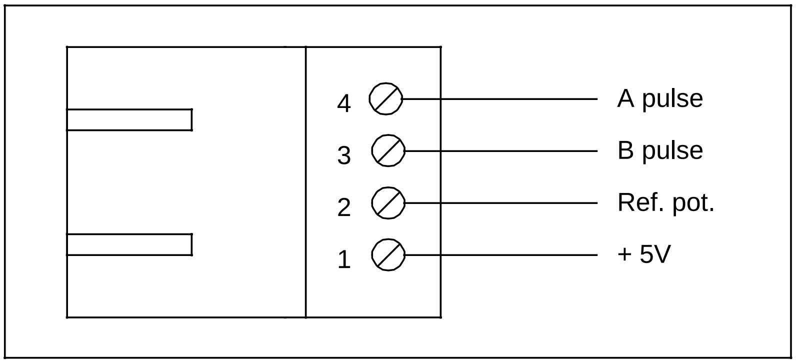


Characteristics:

- RS232C: (V.24) Level ± 12 V
  Signals *RxD and *TxD are low active.
- 20 mA: Active or passive determined in connector.

7.5 Handwheel interface module (6FX1 126-5AA.)

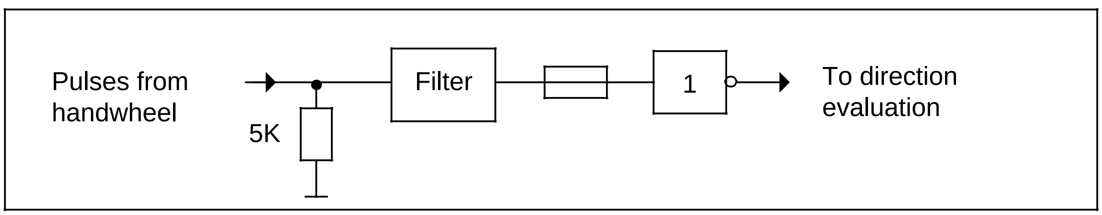
7.5.1 Connector pin assignment



The control can supply max. 150 mA per handwheel at the +5V pin.

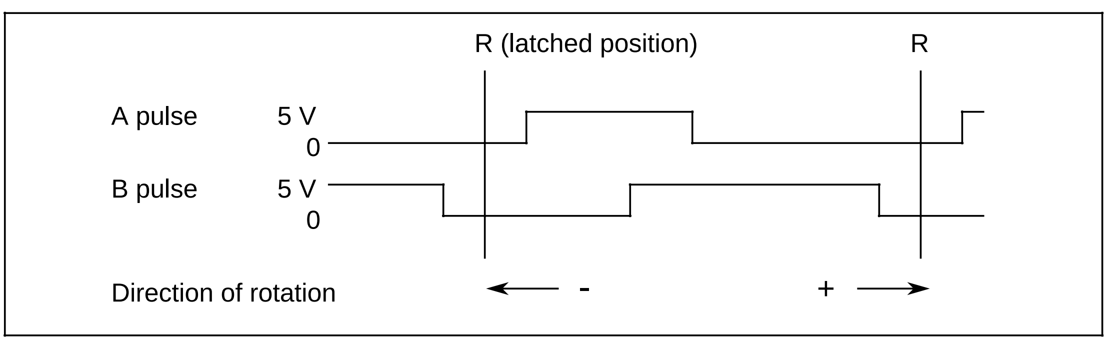
The handwheel voltage supply and the pulses from the handwheel are routed via a 4-pin connector.

7.5.2 Equivalent circuit diagram



2 pulses A and B with 90° offset are output when changing from one latched position to the next (pulse/no-pulse ratio 1:1).

The outputs are at low level in the latched positions.



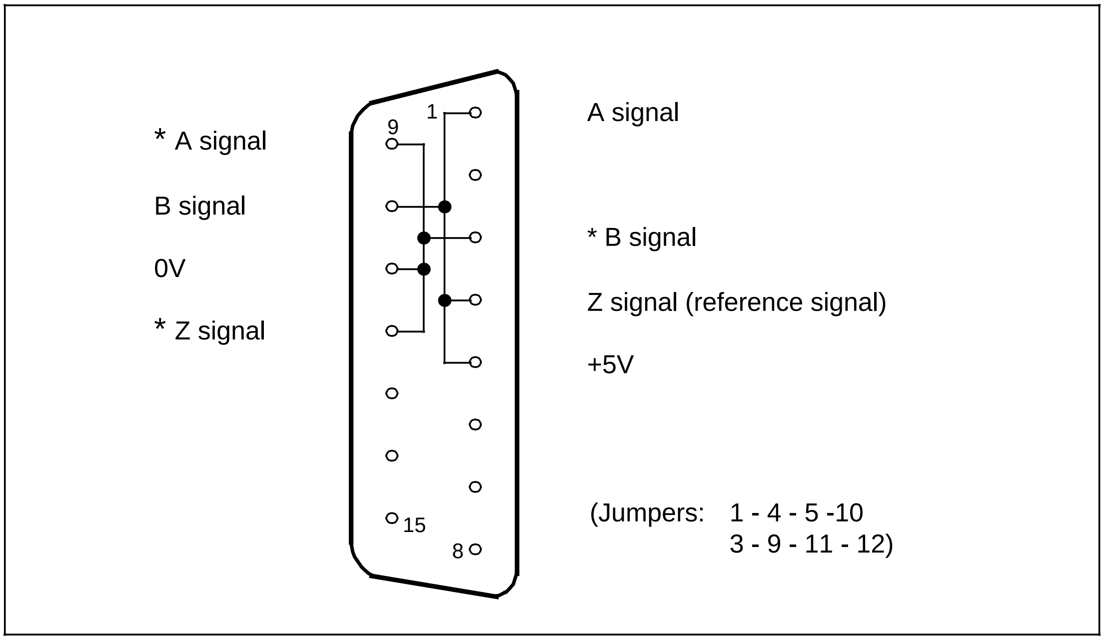
Max. frequency	5 kHz
Low level	0.9V
High level	3.6V

7.6 Short-circuiting connector for actual-value input

The short-circuiting connectors for System 3 and System 8 digital measuring systems must **not** be used for the System 800 in view of the different pin assignment.

The connector is used to test the system and measuring circuits with no encoders connected or with no axis available if a modification to MD 560* bit 0 is not advisable.

User manufacture of connector:



8 NC Machine Data

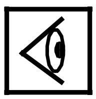
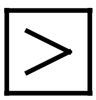
8.1 General

MDs are used to match the NC to the machine tool.

They should be carefully determined and optimized at the time of installation unless the machine manufacturer or end user specifies fixed settings.

Input is in the RAM memory on the 6FX1 138-5B... CPU module (MD card), which is battery-protected against loss of data should the NC be disconnected. The battery itself is not contained on the module.

The following sequence of operations must be observed when storing or manually modifying the MD:

1. Key 
2. Password ”. . . . . ”
3. Softkey: ”DIAGNOSIS”
4. Key 
5. Softkey: ”NC COMM”
6.

GENERAL DATA	AXIAL DATA 1	AXIAL DATA 2	SPINDLE DATA	MACHINE BITS	>	CHANNEL DATA

NC MD	0 ... 261:	General values
NC MD	1080 ... 118*:	Channel-specific values
NC MD	200* ... 396*:	Axis-specific values
NC MD	4000 ... 461*:	Spindle-specific values
NC MD	5000 ... 5050:	General bits
NC MD	5060 ... 5066:	Transmit values
NC MD	5200 ... 521*:	Spindle-specific bits
NC MD	540* ... 558*:	Channel-specific bits
NC MD	560* ... 584*:	Axis-specific bits
NC MD	6000 ... 6249:	Leadscrew error compensation bits
NC MD	1096*:	Axis-specific values
PLC MD	0 ... 9:	General system data
PLC MD	1000 ... 1007:	PLC user MD
PLC MD	2000 ... 2005:	General system bits
PLC MD	3000 ... 3003:	PLC user MD BITS

Effectiveness of individual MDs:

- Configuration-specific data .... After Power On
  (e.g.: MD 200*, 4000, 156, ...)
- Axis-specific data .... After RESET
  (e.g.: MD 204*, 240*, ...)
- Spindle-specific data .... After NC START
  (e.g.: MD 5201, bit 1, ...)
- Display-specific data .... Immediate
  (e.g.: MD 5007, bit 7, ...)

In case of doubt it is advisable to execute the Power On routine as **all** MDs are activated (with the exception of MD 5, 8, 12, 5015 bit 6).

This is applicable when

a) the NC is switched off and on and when warm restart has not been selected
   or

b) after MD input the following softkeys are actuated and warm restart has been selected:
   1. ”INITIAL CLEAR” and
   2. ”SET UP END PW”

The memory must be reformatted after certain MDs have been modified.

- MD 5 ”FORMAT USER MEM.” softkey and
  ”CLEAR PARTPR.” softkey
- MD 8 As for MD 5
- MD 5015 bit 6 ”FORMAT USER MEM.” softkey

INPUT UNITS

MS unit= 2 units of position control resolution (MS reference system) MS: measuring system

e.g. 1 unit of position control resolution=1/2 µm (MD 584*=xxxx0010)
x .. no significance here

1unit (MS)=1 µm

IS unit= 1 unit of input resolution (IS reference system) IS: input system

e.g. 1 unit of input resolution=1 µm (MD 5002=010xxxxx)
x .. no significance here

1unit (IS)=1 µm

VELO ... smallest unit of digital-analog converter (DAC) for setpoint conversion

Given a 14 bit DAC then: $1\ \text{VELO} = \frac{10\text{V}}{8192} = 1.22\ \text{mV}$

Display resolution

The display resolution is specified in MD 584*

Example: Display resolution (MD 584* = 010xxxxx)
1 unit = 1 µm x... no significance here

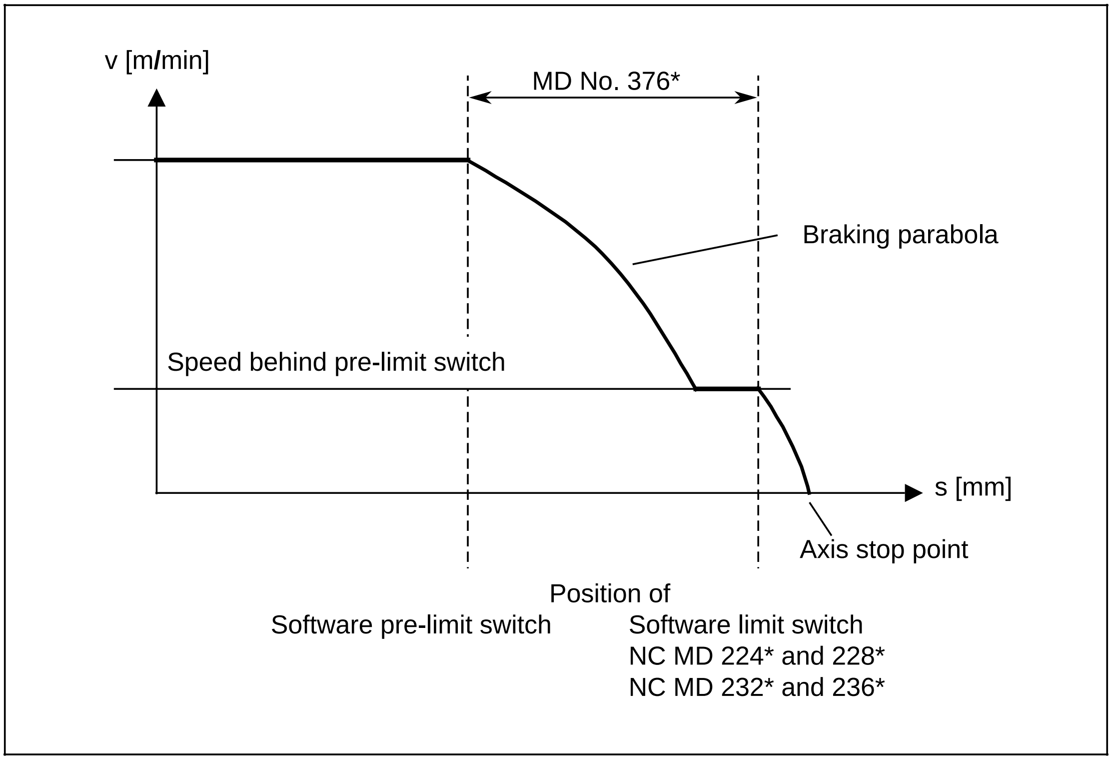
8.2 Description of the NC MD values

Note: MD 0 has been replaced by the axis-specific MD 376*. This allows prelimit switch values to be specified for each axis.

1	Speed behind pre-limit switch		
Standard value	Lower input limit	Upper input limit	Units
500	0	15 000	1 000 units/min (IS)

Note:

MD No. 1 has no effect if zero is entered in MD No. 376*.

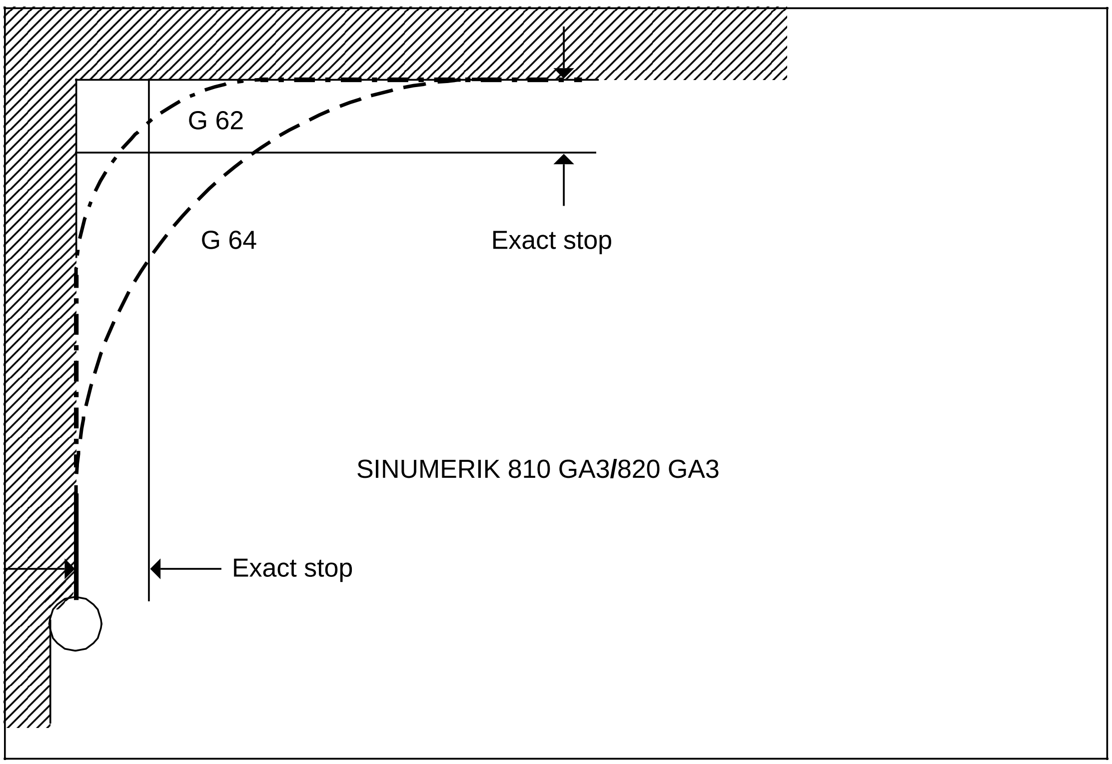


3	**Corner deceleration rate**		
Standard value	Lower input limit	Upper input limit	Units
500	0	15 000	1 000 units/min (MS)

In continuous-path operation (G64) block transitions are covered without feedrate reduction, i.e. the tool path feedrate is maintained and the exact stop tolerance ranges are not scanned.

As a result of function G62 (continuous-path operation with feedrate reduction) the tool path feedrate at the block transition is reduced to the rate entered in MD No. 3, provided the selected feedrate was greater.

The radiusing is thus reduced at discontinuous block transitions.



Schematic of exact stop for corner deceleration

5	Number of MIB parameters		
Standard value	Lower input limit	Upper input limit	Units
250	100	250	-

MIB – **M**achine **I**nput **B**uffer

The MIB parameters are required for input of variables for the cycles and blueprint programming, e.g.

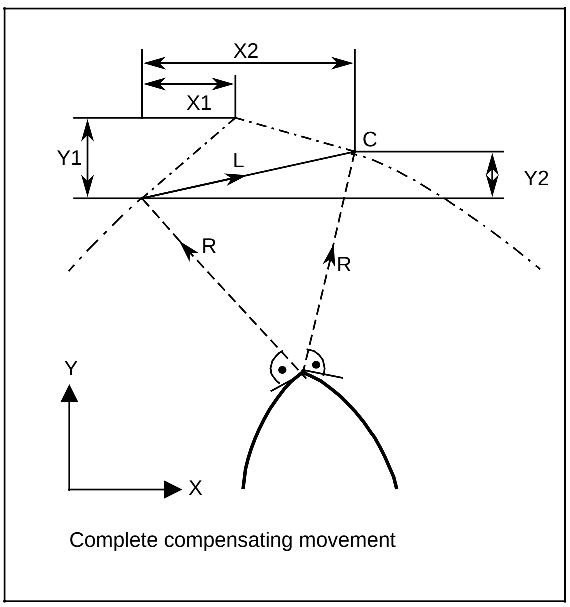
- Reference plane
- Drilling/boring depth
- Finishing allowance
- Chamfer
- Configured displays

100 MIB parameters are used with the displays currently available.

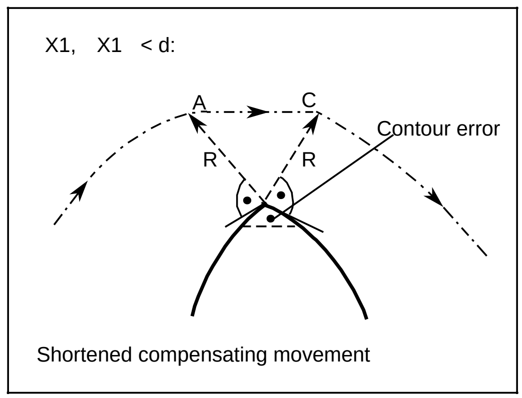
Any modification to this value is not valid until the ”FORMAT USER MEM.” and ”CLEAR PARTPR.” softkey functions have been performed.

6	Threshold for CRC insertion blocks		
Standard value	Lower input limit	Upper input limit	Units
0	0	2 000	units

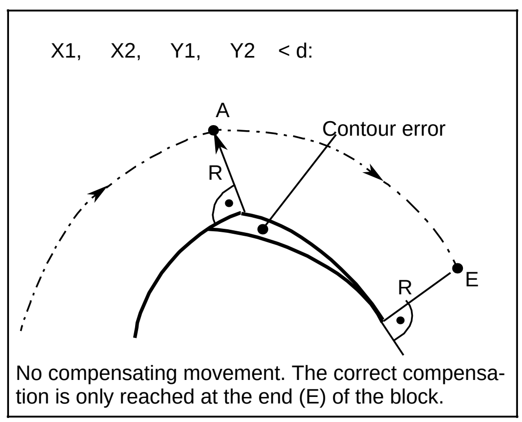
One or more intermediate blocks for linear compensating movement(s) are inserted for transitions from a circular contour to a straight contour or to another circular contour (see Programming Instructions). With these compensating movements, the programmed feedrate is maintained along the cutter centre path; during machining, however, the rate of feed is maintained with respect to the workpiece contour. This results in differences in feedrate. In order to prevent drops in speed if the travel is inadequate, the compensating movements beneath the threshold for CRC insertion blocks are shortened or omitted as follows:



Complete compensating movement



Shortened compensating movement



No compensating movement. The correct compensation is only reached at the end (E) of the block.

X1	Length of 1st compensating movement in X direction
Y1	Length of 1st compensating movement in Y direction
X2	Total length of both compensating movements in X direction
Y2	Total length of both compensating movements in Y direction
d	MD 6 (threshold for CRC insertion blocks)
A	End point of 1st block with CRC selected
R	Cutter radius

7	**Circle end position monitoring**		
Standard value	Lower input limit	Upper input limit	Units
5	0	32 000	units (IS)

Before a circular block is processed, the NC checks the ”correctness” of the programmed values by determining the difference in radii for the starting and end positions. If the difference exceeds the upper limit specified above, the block is not cleared for processing. Alarm 2048 (circle end point error) is displayed.

If the difference is less than MD 7 but not equal to zero, the centre point parameters are corrected since it is assumed that the end position has been correctly programmed. The circle is then traversed on the basis of the new centre point. A threshold value for activation of the correction can be defined in MD 225.

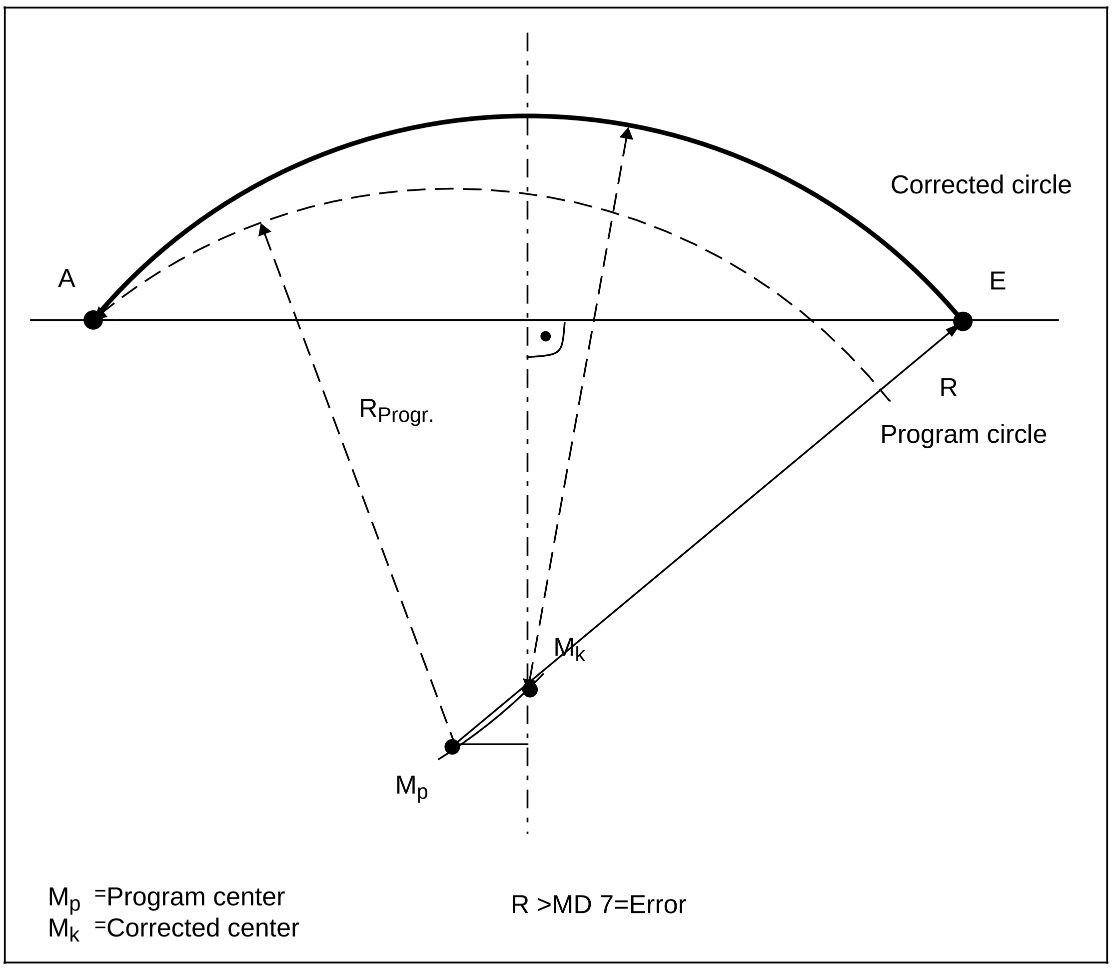


Diagram of circle end point monitoring

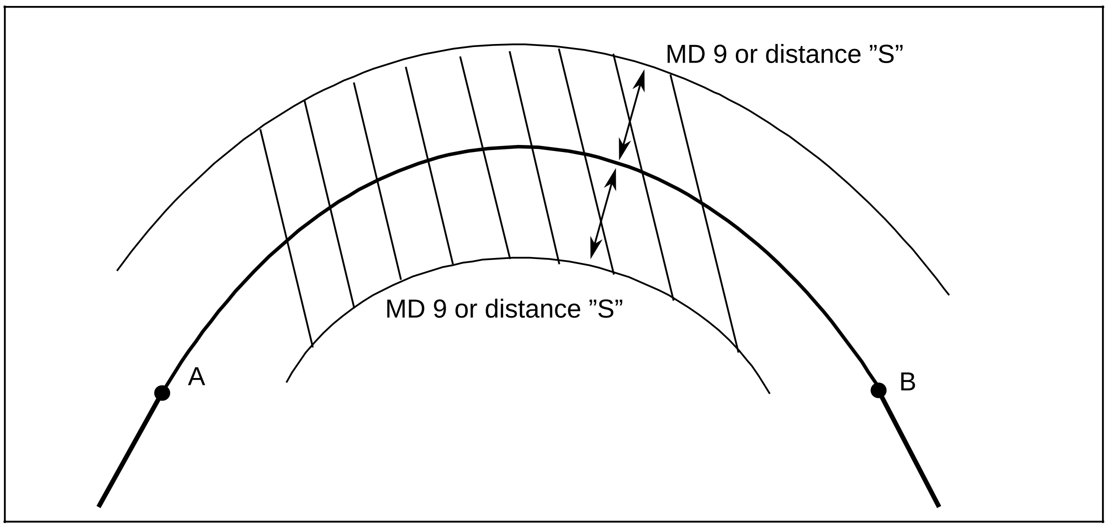
8	Maximum number of machining programs		
Standard value	Lower input limit	Upper input limit	Units
50	0	500	-

For each machining program provided, the SINUMERIK reserves a small memory area (10 bytes) for program organization purposes. 10.000 programs and subroutines can be processed but far fewer programs are usually stored in the memory at any one time (500 max.), so a significant memory area would have to remain empty. To prevent this, matching to the actual part programs anticipated can be performed.

Any modification to this value is not valid until the ”FORMAT USER MEM.” and ”CLEAR PARTPR.” softkey functions have been performed.

9	Error window for repositioning on circle contour		
Standard value	Lower input limit	Upper input limit	Units
200	0	32 000	units (IS)

If, in AUTOMATIC mode, the traversing movement has been terminated (NC STOP) in a circular block with possible retraction in JOG mode from the interrupt point, the distance between the axis and the interrupt point is checked following reapproach to the interrupt point on NC START. The control considers two distances for the maximum permissible distance to the circle contour, MD 9 and distance ”S” resulting from the formula given below. The smaller of the two distances is chosen. If the distance to the circle contour is larger than the smaller of the two distances, the program is not activated and alarm 3018 (distance from contour too great) is set.



The actual axis position must be in the hatched area on NC START.

Formula: $S = \frac{\text{MD 276* } \cdot \text{ IPO}^2}{200}$ (units)

Example: Calculation of the distance for repositioning

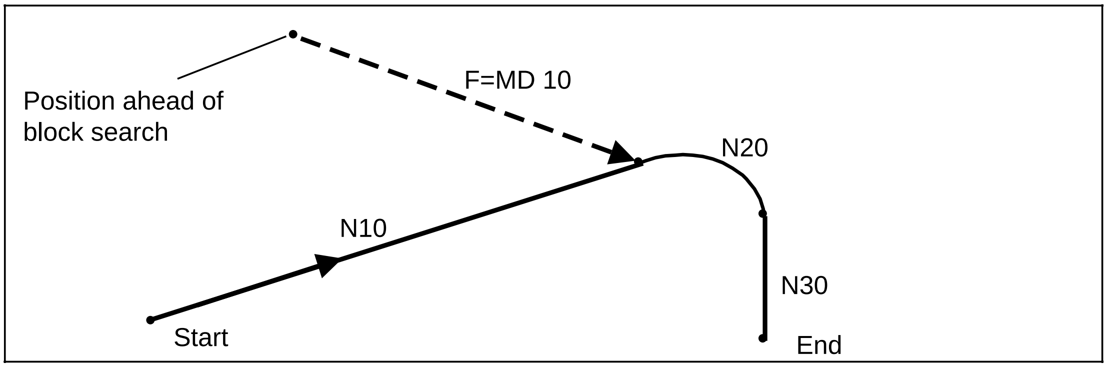
NC MD 276*: 200
NC MD 9: 200
IPO frequency: 20 ms

$S = \frac{200 \cdot 20^2}{200} = 400$ (units)

Formula S = 400 not active because of limitation to **smaller** distance in NC MD 9!

10	Feed after block search		
Standard value	Lower input limit	Upper input limit	Units
1 000	0	15 000	1000 units/min (IS)

Rate of return to contour after block search.
If the machining program is not initiated at the start but at an existing block using block search, the programmed rate may not be suitable for the travel (e.g. F value very small). This is why the speed specified in NC MD 10 is used to traverse to the end point of the selected block after block search. The speed which resulted from the block search is used to continue.



Caution:

MD 10 has no effect with block search for a block with G00 (rapid traverse movement), instead rapid traverse is used to traverse to the end point of the block that has been input.

11	Password		
Standard value	Lower input limit	Upper input limit	Units
0	0	9 999	-

A four-place password (in BCD code: 0 to 9999) can be entered in the common NC MD11 and used for comparison every time a password is requested under the following conditions.

- When MD 11=0 the password is ”1111” as it has been until now.
- Leading zeros must be entered with the password.
- In normal mode (when the password has not been entered) NC MD 11 is not displayed.

13	Number of TO parameters		
Standard value	Lower input limit	Upper input limit	Units
10	8	10	-

The SINUMERIK 810 GA3/820 GA3 has 99 tool offsets with 8 or 10 TO parameters (P0, P1, ... P9).

The significance of the individual TO parameters is adequately described in the Programming Guide (e.g. P1=TO type).

14, 15 16, 17	Protected R parameters (cycle machine data and setting data)		
Standard value	Lower input limit	Upper input limit	Units
10 000	0	10 000	-

Appropriate definitions in MD 14 - MD 17 make it possible to protect R parameter ranges from input via the keyboard. In this way cycle machine data and setting data (cycle MD/SD) can be implemented.
The range is specified via general NC machine data.

- Cycle MDs: MD 14 from R parameter number
  MD 15 to R parameter number
  Input disabled by means of password

- Cycle SDs: MD 16 from R parameter number
  MD 17 to R parameter number
  Input disabled by means of keyswitch (activated by NC MD 5005, bit 3)

If the ranges overlap the corresponding data are defined as cycle SD.

20	Size of circular buffer		
Standard value	Lower input limit	Upper input limit	Units
0	0	64	kByte

The circular buffer has a minimum size of 1 Kbyte and this size at least must be set when the ”Block transfer” option is active. The size of the circular buffer can be set in Kbytes via NC MD 20, the upper limit being 64 Kbytes.

The circular buffer is set up in the part program memory when the part program is formatted. This means that the circular buffer reduces the memory area for part programs occupying it completely in the extreme case. Whenever the size of the circular buffer is to be changed, this must be entered in NC MD20. The part program memory must then be reformatted.

The circular buffer can accommodate a maximum of 2 part programs. This provides the following possibilities:

One part program or part of one part program is accommodated in the circular buffer, classical BTR operation, or two part programs are contained in the circular buffer which are executed consecutively with NC Start. The first program can be executed while the second program is being read in. After the first program has been completed, the second program can be started and a third program can be read in. This sequence can be determined arbitrarily. If this procedure is used in the part program memory and not in the circular buffer, the part program memory will be full after a certain period of time and a Reorg must carried out. This is not necessary in the circular buffer, because each executed program is declared invalid and thus memory capacity is available for the next program transferred.

100 130	Positions 2 to 32 of feedrate override switch		
Standard value	Lower input limit	Upper input limit	Units
-	0	150	%

Use can be made of a feedrate override switch with up to 32 positions. The % figures may be allocated as required, only the far left switch position (position 1) being specified at 0%. If 0% is allocated to another switch position, the feed hold LED does not light up, unlike in position one.

The following standard values are entered with automatic MD setting:

1, 2, 4, 8, 10, 20, 30, 40, 50, 60, 70, 75, 80, 85, 90, 95, 100, 105, 110, 115, 120, the remaining MD (MD 122 – MD 130) are set to 0.

131 146	Positions 1 to 16 of spindle override switch		
Standard value	Lower input limit	Upper input limit	Units
-	50	130	%

Assignment to max. 16 spindle override switch positions as required.
The following standard values are entered with automatic MD setting:

50, 55, 60, 65, 70, 75, 80, 85, 90, 95, 100, 105, 110, 115, 120, 120.

MD 146 is not evaluated.

147 154	Positions 1 to 8 of rapid traverse override switch		
Standard value	Lower input limit	Upper input limit	Units
-	0	100	%

The standard values are: 1, 10, 50, 100.

155	Setting of position controller sampling interval		
Standard value	Lower input limit	Upper input limit	Units
2	0	5	0.50 ms

The sampling interval for the position controller is adjustable from 4 to 6.5 ms (i.e. a new set speed can be output every 5 ms). The basic value for the sampling interval is 4 ms. It can be increased by the amount of the input in 0.5 ms increments, e.g. an input value of 3 yields a new sampling interval of 5.5 ms.

156	Servo enable cutoff delay		
Standard value	Lower input limit	Upper input limit	Units
200	0	16 000	ms

The speed controller enable (servo enable) at the measuring circuit is cancelled after the set delay has elapsed. The servo enable is provided on the measuring circuit once for each axis spindle.

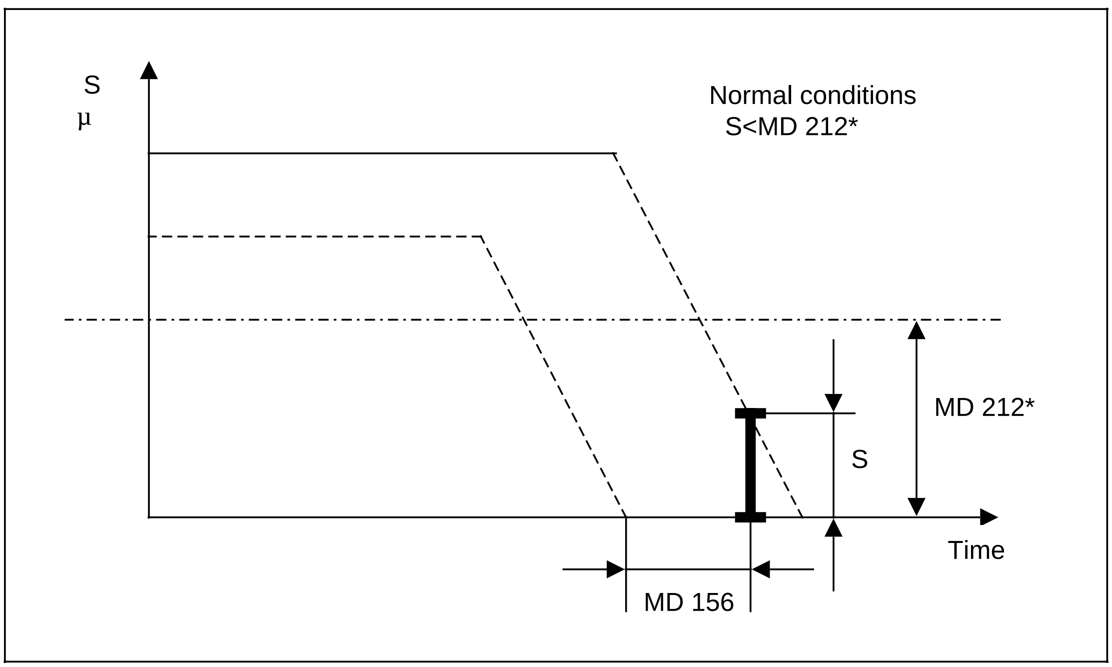
The effect of the delay input is as follows:

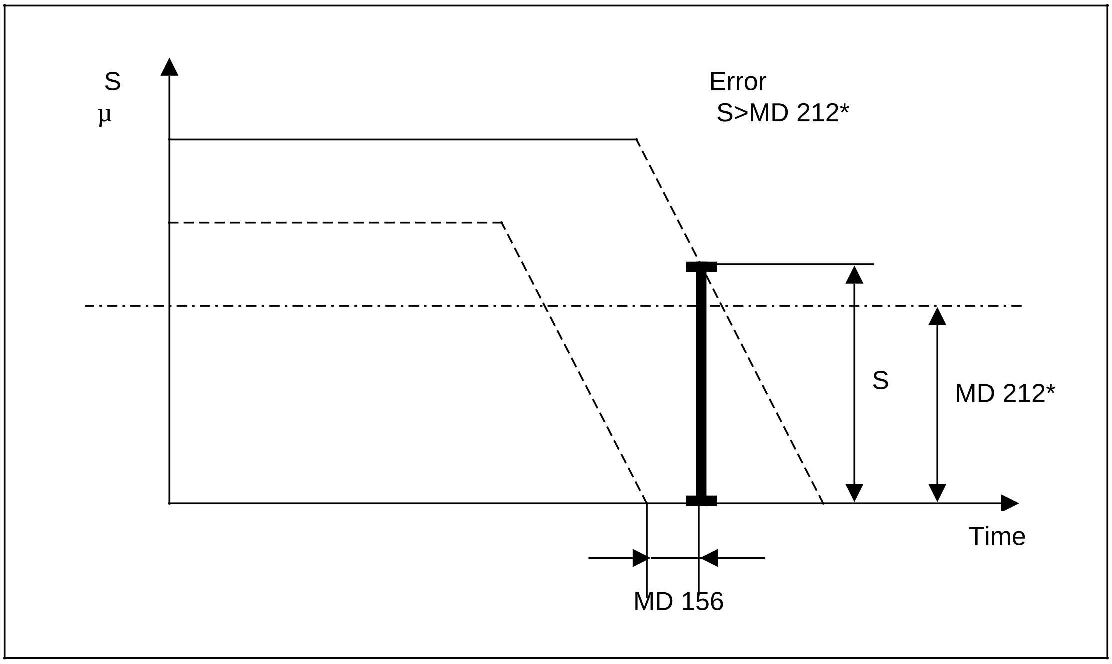
1. Once the interpolator has reached the programmed position, the clamping tolerance (MD 212*) is activated after this delay has elapsed. At this instant the following error must therefore be less than the clamping tolerance. The delay selected should be such that the maximum following error (rapid traverse) can be suppressed. In the event of an error, the servo enable is cancelled and alarm 112* (zero-speed control) is activated.

   This applies only if NC MD 372* (delay zero-speed control) is set to 0.

2. Delay for cancellation of servo enable at the measuring circuit after ”EMERGENCY STOP” and other errors which result in immediate shutdown of the axes (e. g. contour monitoring, zero-speed control).
3. Delay for cancellation of servo enable at the measuring circuit if the interface (PLC) cancels the servo enable signal for a traversing axis.

Any modification to MD 156 is only valid after POWER ON.





157	Control types for standard cycles		
Standard value	Lower input limit	Upper input limit	Units
see possible values	0	9 000	-

On all controls of the System 800, machine data "MD" 157 must be kept free. On Power On this MD is overwritten by the software. It is used to identify the control type and software version. This data is read cyclically to ensure compatibility of all controls from SINUMERIK 805 to SINUMERIK 880.

Type of control	MD 157
810 T	11 xx
810 M	12 xx
810 G	13 xx
820 T	21 xx
820 M	22 xx
850 T	51 xx
850 M	52 xx
880 T	81 xx
880 M	82 xx

The Software Version is coded in the two places xx (see example).

Example:

SINUMERIK 810M with NC Software Version 2:
MD 157 = 1220

Software version — Release of the software version.

200 223	Key assignments		
Standard value	Lower input limit	Upper input limit	Units
0	0	see table	–

In the SINUMERIK 810 T/M GA3 SW 3, new characters can be assigned to the keys on the central keypad by means of the machine data 200 to 223. The new key assignments become effective after POWER ON.

The key designation must be specified in decimal code as follows:

MD 2xx: nnn aaa

aaa: Coding of the key without shift function

nnn: Coding of the key with shift function

Example for MD 200: 037 097 ⟹ a / %

037: %

097: a

Following the function ”End of installation”, the standard assignments apply for the keys, i.e. machine data MD 200 to 223 are then provided with the relevant values (see Installation Lists).

Assignments of machine data to the dual function keys on the keypad. The position of the machine data corresponds to the key position on the keypad. The shift key (position ”-”) cannot be altered.

MD 200	205	210	215	–
MD 201	206	211	216	220
MD 202	207	212	217	221
MD 203	208	213	218	222
MD 204	209	214	219	223

Valid characters/code

Dec. code	ASCII	Meaning	Dec. code	ASCII	Meaning
10		Line feed	71	G	
32		Blank	72	H	
33	!	Exclamation mark	73	I	
34	”	Quotation marks	74	J	
35	#	Number sign	75	K	
36	$	Dollar sign	76	L	
37	%	Percent sign	77	M	
38	&	Commercial AND sign	78	N	
39	'	Apostrophe	79	O	
40	(	Left parenthesis	80	P	
41	)	Right parenthesis	81	Q	
42	*	Asterisk	82	R	
43	+	Plus	83	S	
44	,	Comma	84	T	
45	–	Minus	85	U	
46	·	Point	86	V	
47	/	Slash	87	W	
48	0		88	X	
49	1		89	Y	
50	2		90	Z	
51	3		91	[	(for 810 only)
52	4		92	\	Backslash
53	5		93	]	(for 810 only)
54	6		94	^	Circumflex
55	7		95	_	Underscore
56	8		96	`	Grave accent
57	9		97	a	
58	:	Colon	98	b	
59	;	Semicolon	99	c	
60	<	Less than	100	d	
61	=	Equal to	101	e	
62	>	Greater than	102	f	
63	?	Question mark	103	g	
64	@		104	h	
65	A		105	i	
66	B		106	j	
67	C		107	k	
68	D		108	l	
69	E		109	m	
70	F		110	n	

Dec. code	ASCII	Meaning	Dec. code	ASCII	Meaning
111	o		140		Delete one character in the LEB line
112	p		141		Select
113	q		142		Page up
114	r		143		Page down
115	s		144		Cursor one field backwards
116	t		145		Cursor one field forwards
117	u		146		Cursor one field up
118	v		147		Cursor one field down
119	w		148		Large actual value display on/off
120	x		149		Alarm acknowledgement (cancel))
121	y		150		Installation/start-up
122	z		151		Menu on/off
123	{	Left brace	152		No meaning
124	|	Vertical line	153		Channel changeover
125	}	Right brace	154		No meaning
126	~	Tilde	155		”
127	DEL	Delete character	156		”
128		1st softkey	157		”
129		2nd softkey	158		Back to initial menu
130		3rd softkey	159		No meaning
131		4th softkey	160		1st axis (letter of axis)
132		5th softkey	161		2nd axis ”
133		ETC key	162		3rd axis ”
134		RECALL function	163		4th axis ”
135		Invert sign	164		Radius
136		Shift on/off	165		Angle
137		Input key			
138		EDIT			
139		CANCEL word/block			

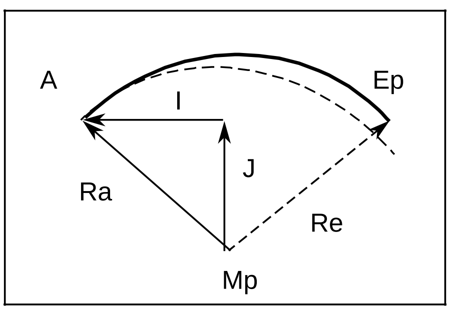
A decimal code of > 165 is not allowed. The key code must be entered as a 3-digit decimal number for each key page.

If an error occurs, the following alarm message is issued:

88 1 N 200 Illegal key code

- 200 — MD No. in which the incorrect key code is to be found
- 1 — 1 = Error in key (without shift)
  2 = Error in key (with shift)
- 88 — Error number

225	Threshold value circle centre offset		
Standard value	Lower input limit	Upper input limit	Units
0	0	1000	IS



Example:
:
N61 G01 X(A) Y(A) F100 LF
N62 G02 X(E) Y(E) I20 J20 F50 LF
:
A Starting point
Ep Programmed end point
Ra=Radius at starting point
Re=Radius at end point

If the value in MD7 > !Ra-Re!   MD225, the circle centre (determined, for example, by the programmed parameters I and J) is shifted such that the circular section to be traversed passes through the point Ep. With !Ra-Re!<MD255 the circle centre is not shifted; instead, the program end value Ep is approached at the end of the circular motion.

250	Language selection		
Standard value	Lower input limit	Upper input limit	Units
0	0	1	–

The NC is supplied as a bilingual version. The first language is selected by value 0.

The second language is selected by value 1.

The MD 250 applies after a general reset.

Examples of language combinations:

Language 1	Language 2
German	English
German	French
German	Italian
German	Spanish
English	French
English	Italian
English	Spanish

The two languages only refer to the texts in the NC and not to texts stored in the UMS.

In the ”Software version display” diagnostics display, the presently preselected language is marked by means of the arrow ”>”.

260	Selection rotary axis mode		
Standard value	Lower input limit	Upper input limit	Units
–1	–1	99	–

This machine data defines the M function for selecting the rotary axis mode from the spindle mode. Values between 6 and 99 can be used but M0, M01, M02, M03, M04, M05, M17, M19, M30, M36, M37 must not be specified as this would cause Alarm 2184 to be output. The value ”-1” prevents the rotary axis mode from being selected by means of the M function.

261	Deselection rotary axis mode		
Standard value	Lower input limit	Upper input limit	Units
–1	–1	99	–

This machine data defines the M function for deselecting the rotary axis mode. The number range of MD 260 is allowed, but MD 261 must be different from MD 260. Furthermore, the M function M05 must be used in the case of MD 261.

Example: MD 260 = 70
MD 261 = 71

then, for spindle 1: Selection of rotary axis mode M1 = 70
Deselection of rotary axis mode M1 = 71

where the extension of M specifies the spindle number. The value ”-1” prevents deselection of the rotary axis mode by means of M function.

Channel-dependent machine data

In the past a reset position was predetermined for the G functions contained in G groups 1, 3, 4, 6, 8, 10, 11, 12 and 14. As from software version 2 and as from channel-specific machine data 108*, the reset positions must be specified per channel for G groups 1, 3, 6, 8 and 12. The reset position for G70/71 continues to be preset in MD 5002 bit 4.

In the following MDs the * must be replaced by "Channel No. 1", e. g. MD 108* means 1080 in channel 1 and 1081 in channel 2.

108*	**Reset position for G group 0**		
Standard value	Lower input limit	Upper input limit	Units
1	0	35	-

Specification of the required G function for the reset position for the 1st G group (G group 0).
Possible input: 0, 1, 2, 3, 6, 10, 11, 12, 13, 33, 34 or 35.

110*	**Reset position for G group 2**		
Standard value	Lower input limit	Upper input limit	Units
for T: 18 for M: 17	17	19	-

Specification of the required G function for the reset position for the 3rd G group (G group 2).
Possible input: 17, 18 or 19.

112*	Reset position for G group 5		
Standard value	Lower input limit	Upper input limit	Units
54	54	57	-

Specification of the required G function for the reset position for the 6th G group (G group 5). Possible input: 54, 55, 56 or 57.

114*	Reset position for G group 7		
Standard value	Lower input limit	Upper input limit	Units
for T: 64 for M: 60	60	64	-

Specification of the required G function for the reset position for the 8th G group (G group 7). Possible input: 60, 62, 63 or 64.

118*	Reset position for G group 7		
Standard value	Lower input limit	Upper input limit	Units
for T: 95 for M: 94	94	97	-

Specification of the required G function for the reset position for the 12th G group (G group 11). Possible input: 94, 95, 96 or 97.

Axis-specific values

40 axes are provided in terms of software to keep the NC system software universal and to be prepared for future developments. However, on the SINUMERIK 810 GA3/820 GA3 only the first 7 axes can be activated.

The significance of the MD changes therefore in increments of 40 MD numbers, e.g.

MD No.	2040	Coarse exact stop for	1st axis
	2041	”	2nd ”
	2042	”	3rd ”
	2043	”	4th ”
	2044	”	5th ”
	2045	”	6th ”
	2046	”	7th ”
	2047 to 2079	Coarse exact stop for (not relevant on SINUMERIK 810 GA3/820 GA3)	8th to 40th axis
	2080	Fine exact stop for	1st axis
	. .	etc.	

The last position of the MD No. is shown by the symbol * (e.g. 204* = coarse exact stop) for the sake of clarity. When the MDs are displayed and input, the actual figure must be inserted for this symbol * as shown above.

*	0	1st	axis
	1	2nd	axis
	2	3rd	axis
	3	4th	axis
	4	5th	axis
	5	6th	axis
	6	7th	axis

200*	Axis assignment		
Standard value	lower input limit	Upper input limit	Units
see possible values	see possible values	see possible values	-

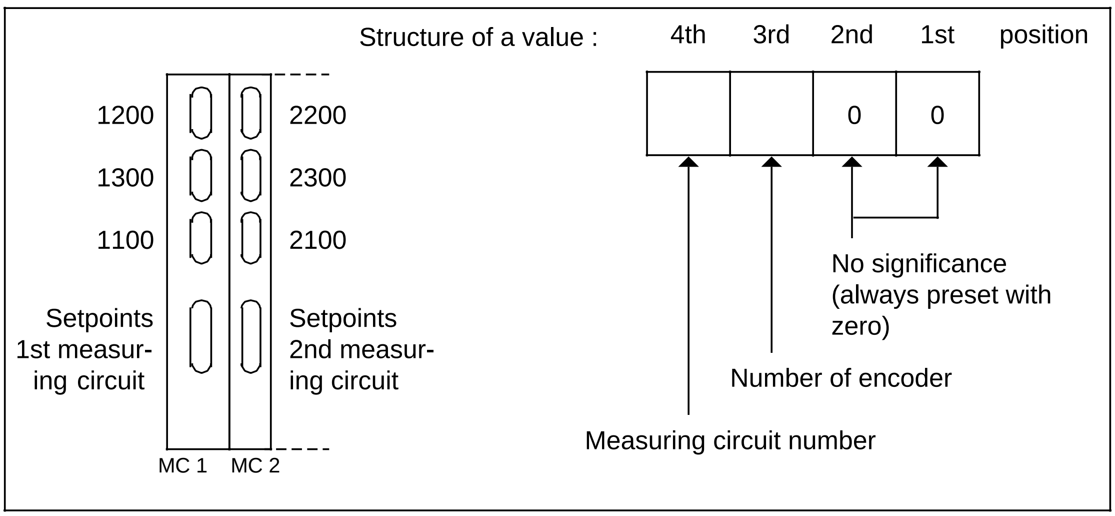
This MD specifies which measuring circuit is to be assigned to which axis, e.g. for milling machines assignment is usually as follows:

Measuring circuit 1 to the 1st axis (X)
Measuring circuit 2 to the 2nd axis (Y) and
Measuring circuit 3 to the 3rd axis (Z)

It defines the measuring-circuit connector to which the actual values and setpoints (command values) for the individual axes are connected.

- **Definition when using the SPC module**

Definition of axis 1:

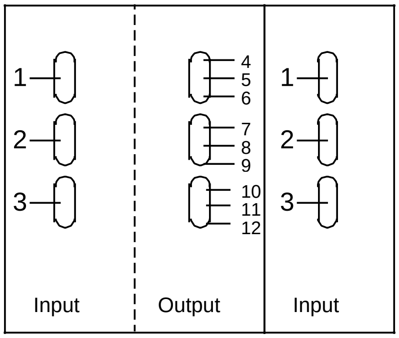


Possible values:

0000	Axis not available at machine or fictitious axis (only permissible with MD 564* bit 7=0 for real axes)
1100	Measuring circuit 1, encoder 1
1200	Measuring circuit 1, encoder 2
1300	Measuring circuit 1, encoder 3
2100	Measuring circuit 2, encoder 1
2200	Measuring circuit 2, encoder 2
2300	Measuring circuit 2, encoder 3

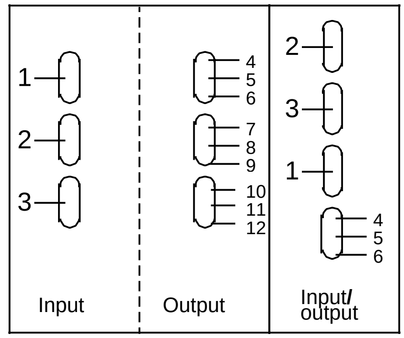
Standard assignment:	810M GA3/		810T GA3/	
	820M GA3:	X axis=1100	820T GA3:	X axis=1100
		Y axis=1200		Z axis=1200
		Z axis=1300		

- **Definition when using the HMS module**



1st HMS meas. circuit (-6BA..)   2nd HMS meas. circuit (-6BB..)

or



1st HMS measuring circuit   2nd SPC module

The measuring circuit interfaces are preset as shown below:

00 02	00 03	00 02	04 12
Number of measuring circuit module Input	Number of measuring circuit connection Input	Number of measuring circuit module Output	Number of measuring circuit connection Output

Consequently, the axis assignment for each axis will be complete if MD 200x is defined for the axis.

Example: MD2002 = 1 0 2 0 2 0 4
Measuring circuit module 1 is 6FX1 145-6BAxx (HMS),
Measuring circuit module 2 is 6FX1 121-4BAxx (SPC).

This means:
– the actual value must be read in from the 2nd connection of the HMS module,
– the setpoint must be output to the 4th connection of the SPC module.

The previous definitions of MD 200x can further be used if the module involved is a SPC module.

The NC recognizes by the number of places whether the previous or the new definition is used.

For fictitious axes value 0 must be entered.

204*	Coarse stop tolerance range		
Standard value	Lower input limit	Upper input limit	Units
40	0	16 000	units (MS)

A larger value may be entered in the coarse stop tolerance range than in the fine tolerance range. Consequently, block change to the next machine block is initiated correspondingly earlier.

If this function is not desired, it can be disabled by inputting equal exact stop values in both machine data.

The coarse stop tolerance range is active with the following:

- G00
- Block ahead of G04
- Block ahead of G58/G59/G92/G25/G26
- Block ahead of which only auxiliary functions are programmed
- Single block without G60/G09
- Jog
- Incremental dimension
- Program end

Note:

The stop tolerance limits are not approached in continuous-path mode G64. There is no sequential error as a result of a large number of consecutive positionings since the closed-loop position control is not "shut down" by the exact stop tolerance limit but block 2 is processed ahead of the end position of block 1.

The actual travel is now:

Remainder of block 1 and block 2, etc. If the axis stops for an instant, e.g. because another axis is about to move or because there is no axis movement in this program block, compensation is made so that the following error is 0 and the axis remains precisely in position.

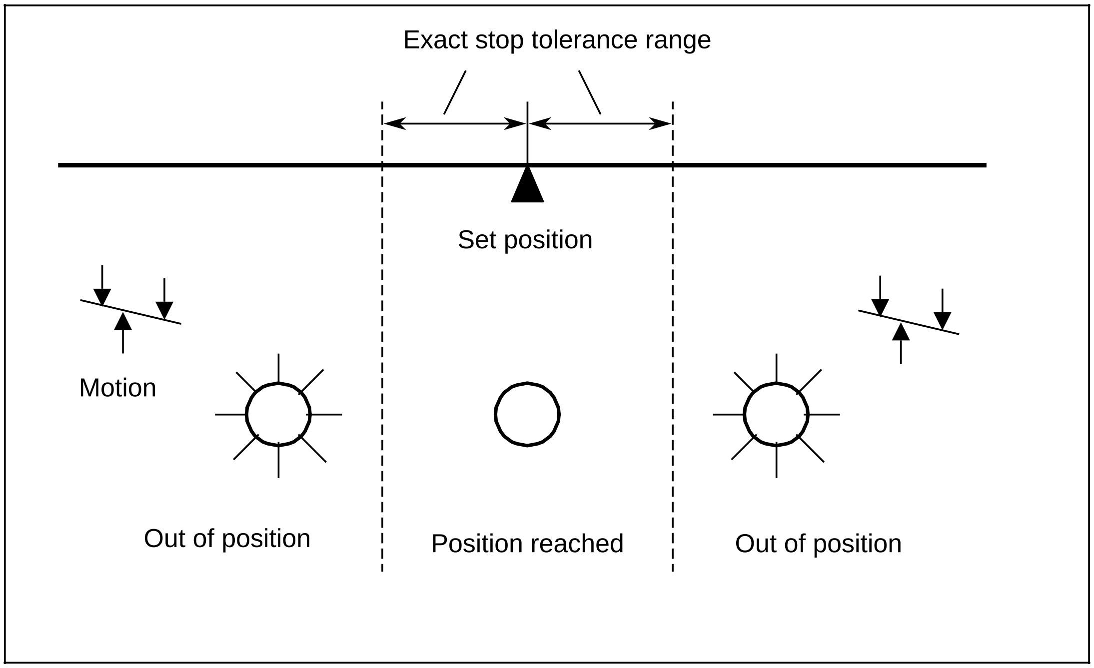
208*	Fine stop tolerance range		
Standard value	Lower input limit	Upper input limit	Units
10	0	16 000	units (MS)

A traversing movement is considered to have been completed when the axis has reached the set position ± the input exact stop tolerance range.

If the actual position is not within this range, the position control light remains on - the block is not then considered completed - and no further movement is possible.

Corrective action:

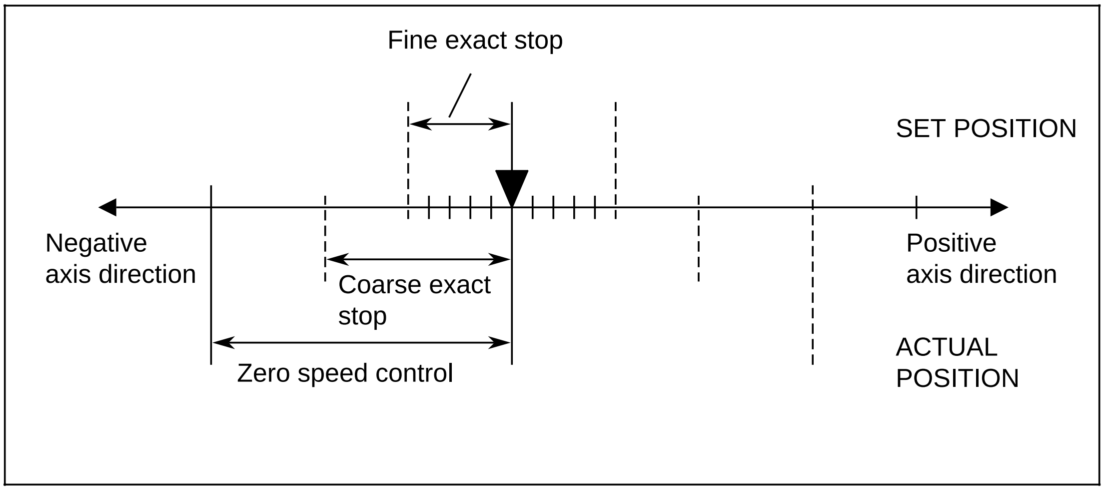
e.g. drift compensation



The fine stop tolerance range is active with:

- G09/G60
- Block ahead of G33

212*	Zero speed control		
Standard value	Lower input limit	Upper input limit	Units
100	0	16 000	units (MS)



The NC monitors the position at rest (holding of position). Alarm 112* is triggered if the zero speed control is exceeded at rest.

The following conditions may occur:

a) If the servo enable signal for an axis is cancelled by the interface controller, this means that the axis can no longer be held in position by the NC. The interface controller must hold the axis in position itself by means of clamping.

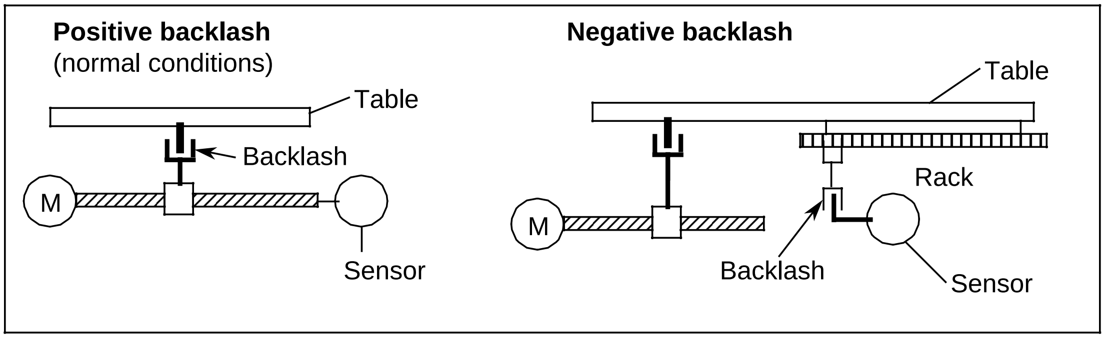
The clamped axis may be forced out of position as a result of mechanical forces.

b) The axis may be forced out of position as a result of major mechanical forces or faults in the drive.

The zero speed control entered must be **greater** than the fine and coarse **stop tolerance ranges**.

220*	Backlash compensation		
Standard value	Lower input limit	Upper input limit	Units
0	0	255	units (MS)

In the case of axes with indirect measuring systems, mechanical backlash results in corruption of the traverse path. On reversal of the direction of movement traverse is either shortened by the amount of backlash or extended, depending on the design.



Actual sensor value ahead of actual value (table):
Table travel too short

Actual value (table) ahead of actual sensor value:
Table travel too far

The correction value entered (amount of backlash) is positive for positive backlash and negative for negative backlash.

224*	Software limit switch 1 (positive direction)		
Standard value	Lower input limit	Upper input limit	Units
99 999 999	- 99 999 999	+99 999 999	units (MS)

The customary range limit switch must be replaced with the software limit switch in the case of the SINUMERIK 810 GA3/820 GA3. The absolute position of the positive range limit for each axis is input in relation to the reference point.

Braking to zero speed is performed so far ahead of the software limit switch that this switch is precisely reached but not overrun (e.g. in jog mode).

Alarm: 148* (Software limit switch positive direction)

Note:

When axes are traversed in interpolation, all axes are shut down if the limit of any of the axes is reached. However, stoppage without contour violation is only guaranteed when MD No. 5003 bit 7 (”No deceleration at limit switch”) is not set, i.e. with braking over the acceleration ramp.

228*	Software limit switch 1 (negative direction)		
Standard value	Lower input limit	Upper input limit	Units
- 99 999 999	- 99 999 999	+99 999 999	units (MS)

Same significance as for MD 224* but for traversing limit in negative direction.
Alarm: 152* (Software limit switch negative direction)

The software limit switches are active only after approach to the reference point of the axis concerned.

Note for MD 224*, MD 228*, MD 232* and MD 236* :

No provision is made for input signals for hardware limit switches. These could only be active via:

- Feed Hold (unfavourable on account of ramp - slow)
- Servo disable (best since fast with jump function)
- Emergency stop (fast with jump function but additional effects, so unfavourable)
  Software limit switches are also effective with fictitious axes.

Note on Transmit:

With active Transmit, the block end points are checked by the NC in the ”AUTOMATIC” modes for software limit switches and working area limitation for the fictitious axes only. The software prelimit switches (see MD 0, MD 1) setting the reduction range take no effect for fictitious axes.

For the real axes in the transformation grouping the following applies: Software limit switches, software prelimit switches and working area limits act on the axes while the axis movement takes place.

If the real axes are outside the limits of the working area or software limit switch and transformation has been selected, traversing into the working area is possible in the fictitious coordinate system. If transformation has not been selected, the real axes can be returned to the working area. Further movement out of the working area/software limit switch range is not possible. With Reset the present alarm message is acknowledged only if the working area limitation or the software limit switch are no longer effective. The transformation is cancelled , if necessary.

232*	Software limit switch 2 (positive direction)		
Standard value	Lower input limit	Upper input limit	Units
99 999 999	- 99 999 999	+99 999 999	units (MS)

A second limit switch position in the positive direction may be specified. Which of the two software limit switches (1 or 2) is then active is selected by the PLC by means of the interface signal (e.g. Q 108 bit 1 for the 1st axis).

Q 108.1	Bit 1=”0”	Bit 1=”1”
	Software limit switch 1 (+) active (1st axis)	Software limit switch 2 (+) active (1st axis)

Example of application:

Reduction of permissible traversing range with tailstock in position.

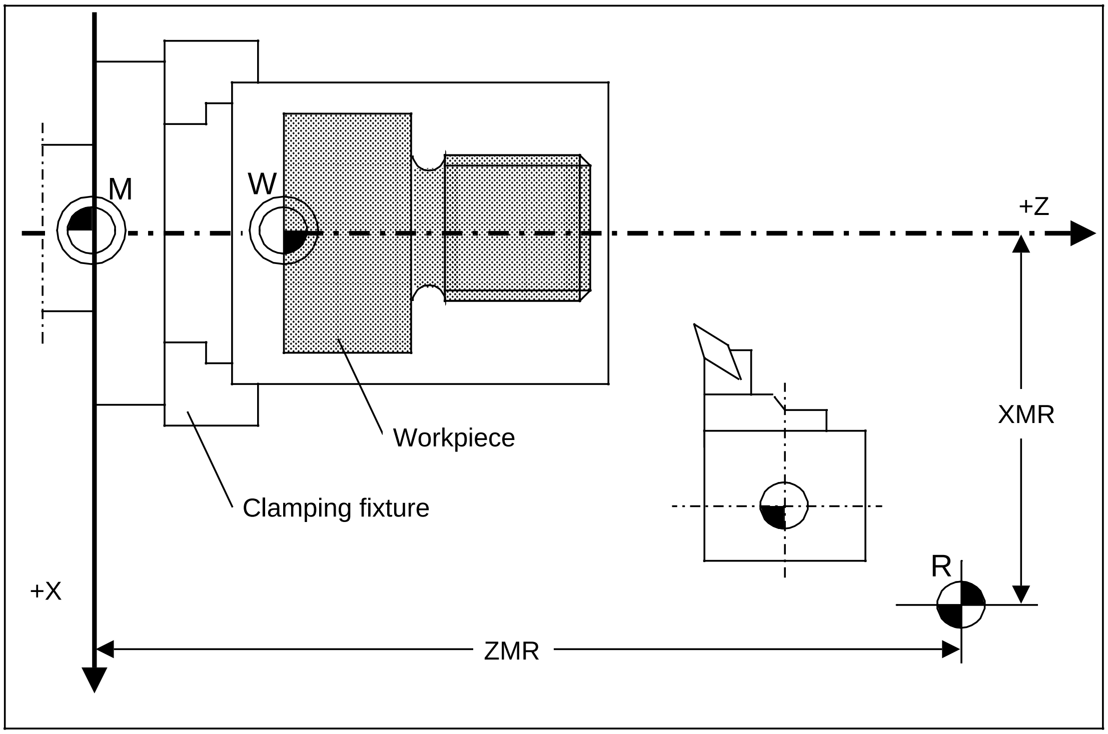
236*	Software limit switch 2 (negative direction)		
Standard value	Lower input limit	Upper input limit	Units
- 99 999 999	- 99 999 999	+99 999 999	units (MS)

Same significance as for MD 232* but in negative direction.
Selection by means of QB 108.0 (for 1st axis).

QB 108.0	Bit 0=”0”	Bit 0=”1”
	Software limit switch 1 (–) active (1st axis)	Software limit switch 2 (–) active (1st axis)

240*	Reference point value		
Standard value	Lower input limit	Upper input limit	Units
0	- 99 999 999	+99 999 999	units (MS)

The difference between the absolute machine zero and the fixed reference point is input for the respective axis. These values are set as actual values for reference point approach.



M . . . . . . Machine zero
W . . . . . . Workpiece zero
R . . . . . . Reference point
XMR . . . . Reference point coordinate in X direction
ZMR . . . Reference point coordinate in Z direction

244*	Reference point shift		
Standard value	Lower input limit	Upper input limit	Units
0	- 9 999	+9 999	units (MS)

The measuring-system reference points may be shifted by means of reference point shift. Consequently, instead of mechanical shifting or rotating of the measuring system (and thus the ”deceleration” cam), the reference point may be shifted electrically up to ±9999 units.

Shifting in excess of the distance traversed between two zero marks is not advisable since this can be achieved by correct adjustment of the operating cam.

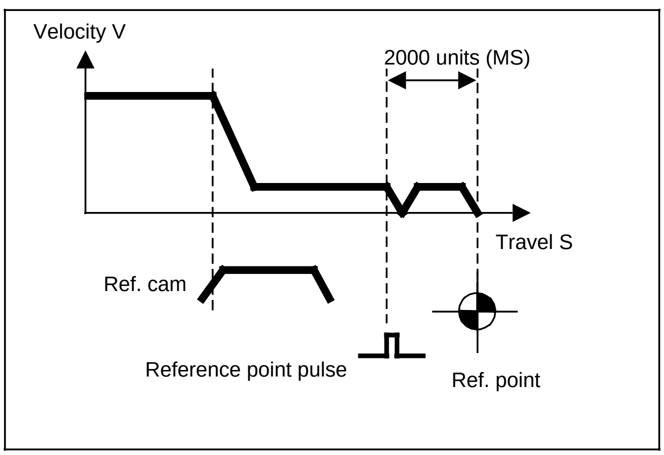
The reference point shift path is traversed at the cutoff (creep) speed (MD 284*) which must already be reached at the operating cam.

Without reference point shift, the reference point is 2000 units (MS) behind the first zero mark after the operating cam has become free again.

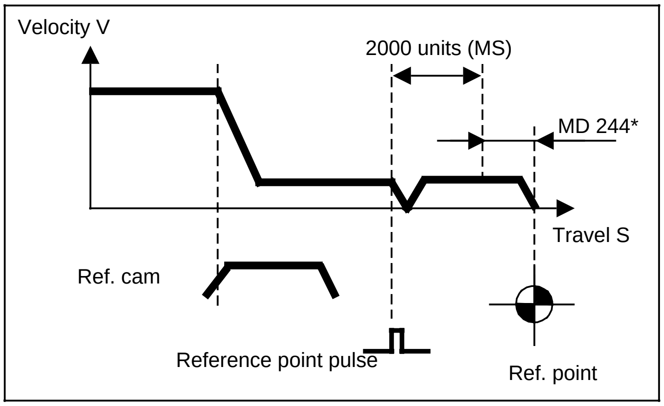
With positive input the axis travels in the positive direction by the input value beyond the normal reference point (2000 units (MS) after zero mark).

With negative input the axis travels, after overrunning the zero mark, to the value resulting from the difference between 2000 units (MS) + the input value. Given a reference point shift of more than approx. 2000 units (MS), the axis reverses in the direction of travel (reverse backlash).

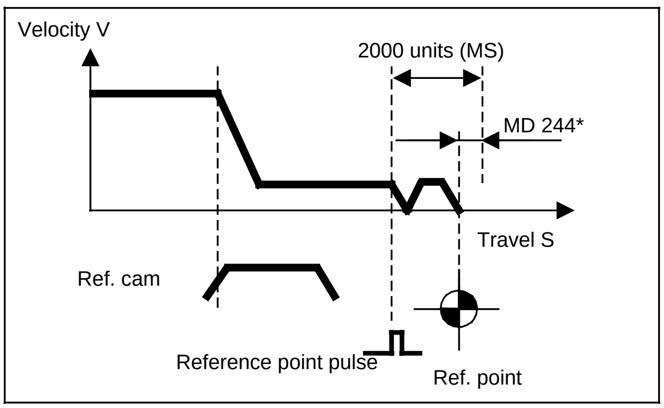
a) MD 244*=0



b) MD 244* greater than 0 (e.g. 1000 units (MS))



c) MD 244* less than 0 (e.g.- 700 units (MS))



248*	**Tool reference value**		
Standard value	Lower input limit	Upper input limit	Units
0	0	±99 999 999	units (MS)

Tool reference value for position of measuring device.

For machines with tool measuring attachments, the reference point of the measuring device relative to the machine zero must be known for automatic determination of the tool geometry data.

Example of application:

Once the tool has been moved into the measuring device, it can be gauged by MD reading with @.

252*	Servo gain factor (Kv factor)		
Standard value	Lower input limit	Upper input limit	Units
1 666	0	10 000	0.01 s^{-1}

When inputting the servo gain factor, it must be borne in mind that the gain factor of the overall position control loop is dependent on other control loop parameters. Strictly speaking there-fore, a distinction must be made between a ”desired servo gain factor” (above MD) and an ”actual servo gain factor” (obtained at the machine). Only if all control loop parameters have been correctly adjusted with respect to each other are these servo gain factors equal. These parameters are as follows:

- Multgain (MD 260*)
- Tacho-generator compensation at speed controller
- Tacho-generator at drive

Note:

Axes to work together in continuous-path operation **must** exhibit precisely the same gain in the position control loop (i.e. at the same speed the same following error = 45° inclination).

Any deviations will result in contour errors!

Only axes **never** contributing to continuous-path operation may be defined with different values.

Example of KV factor setting:

Input resolution:	$1 \cdot 10^{-3}$ mm	MD5002
Position control resolution:	$0.5 \cdot 10^{-3}$ mm	MD5002
Servo gain factor (1):	1666	MD252*
Multgain:	2700	MD260*
Max. speed (10m/min):	10 000 $^{mm}/_{min}$	MD280*

The drive is to be adjusted to 9V corresponding to 10 m/min by means of the battery box. The axis for adjustment is traversed in JOG mode at a speed of 1 m/min. The following error must be observed on the service display.

$$K_V = \frac{\text{Speed}}{\text{Following error}} \left[\frac{\text{m/min}}{\text{mm}} \right]$$

$$\text{Following error} = \frac{\text{Speed}}{K_V}$$

$$\text{Following error} = \frac{1 \text{ m/min}}{1 \ \frac{\text{m/min}}{\text{mm}}}$$

1 mm on the service display corresponds to 2000 units of position control resolution. The set (command) speed should be roughly 737 Velo corresponding to 0.9V (10% of max. speed). In the event of greater deviations with respect to the theoretical following error, the tacho-generator compensation potentiometer at the drive unit must be adjusted. Precision setting of the following error is performed with multgain in view of the better adjustment possibilities.

The axis is then traversed at maximum speed to check the settings. The set drive voltage should be approx. 9V.

The actual servo gain value is 1 if there is a following error of 1 mm at an axis speed of 1 m/min.

256*	**Difference time constant**		
Standard value	Lower input limit	Upper input limit	Units
0	0	+9 999	0.1 ms

With the function tapping with dynamic following error compensation, the dynamic response (inertia) of the axis is adapted to the spindle.

Limits:	0.0 ms	no adaptation
	0.1 ms to 2.7 ms	dead-time element
	2.8 ms to 8.2 ms	P element
	8.3 ms to 999.9 ms	pT2 element

See also Section 10, Description of Operation.

260*	**Multgain**		
Standard value	Lower input limit	Upper input limit	Units
2 400	0	64 000	min/1000 units (MS)

The multgain factor serves to match the controlled system to the servo gain factor specified via MD 252*. Multgain is a strict multiplication factor for the entered servo gain factor and should be used as **precision digital tacho-generator matching** in view of the very fine adjustment facilities. After correct input or matching of the multgain, a servo gain factor corresponding precisely to the input value must be set for the axis concerned.

Matching the actual servo gain factor using the servo gain factor MD (NC MD 252*) is not to be recommended since different input values would then be obtained for the individual axes, although all axes would have the same gain in the position control loop.

Multgain is calculated using the equation: $$\frac{3 \cdot 10^7}{v_{max}} \cdot \frac{U_{max}}{10}$$

where V_{max}: Maximum speed [1000 units/min] [MS]
U_{max}: Maximum voltage at V_{max} [Volts]

Determination of **multgain** as per table:

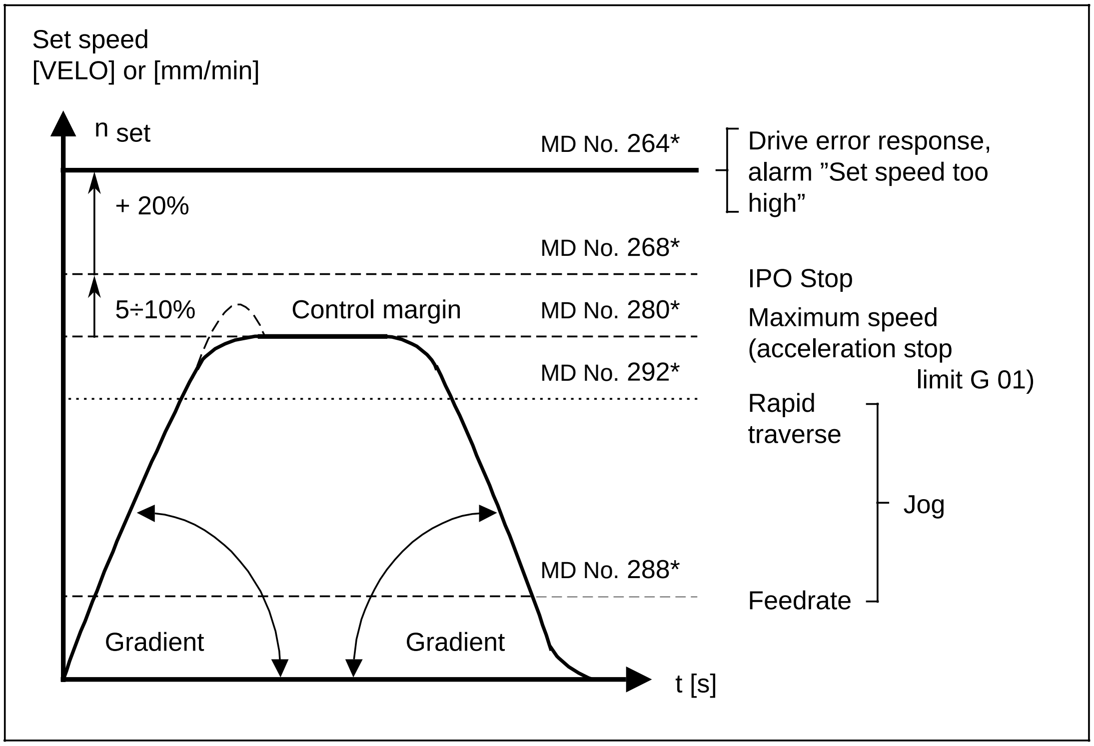
Maximum speed [1000 units/min]	Set speed		
	4 Volts	8 Volts	9 Volts
24000	500	1000	1125
22000	545	1090	1227
20000	600	1200	1350
18000	666	1332	1500
16000	750	1500	1687
15000	800	1600	1800
14000	857	1714	1928
12000	1000	2000	2250
10000	1200	2400	2700
8000	1500	3000	3375
6000	2000	4000	4500
5000	2400	4800	5400
4000	3000	6000	6750
3000	4000	8000	9000
2000	6000	12000	13500
1000	12000	24000	27000
750	16000	32000	36000
500	24000	48000	–
375	32000	–	–
187	64000	–	–

The max. value in **multgain** is 64000.

264*	Drive error threshold		
Standard value	Lower input limit	Upper input limit	Units
9 600	0	15 000	VELO

This monitor triggers Alarm 156* (set speed too high) if the specified set speed is too high. The input value must be greater than the largest definition amount entered in MD 268* for the maximum set speed.

Guide value: Approx. 20% greater than MD 268*



268*	Maximum set speed (IPO Stop)		
Standard value	Lower input limit	Upper input limit	Units
8 192	0	8 192	VELO

This input specifies the maximum voltage value to be output as the set speed. This maximum value depends on any existing setpoint limits in the speed controller (usually 10V). Interpolation Stop (IPO Stop) and Alarm 104* are activated when the limit is exceeded.
The standard machine data of 8192 corresponds to a voltage of 10V.

Caution:

It must, however, be possible to reach the maximum speed (rapid traverse) safely, i.e. tacho-generator compensation is to be performed so that reading and adjustment inaccuracies resulting from speed fluctuations during operation do not result in the IPO Stop limit being reached (e.g. maximum speed = 9 to 9.5V).

272*	**Drift compensation**		
Standard value	Lower input limit	Upper input limit	Units
0	- 500	+500	VELO

The temperature drift of analog electronic components (primarily in the motor control unit) causes the axes to wander from their set position. The counter-setpoint (following error) then corresponds to the current temperature drift.

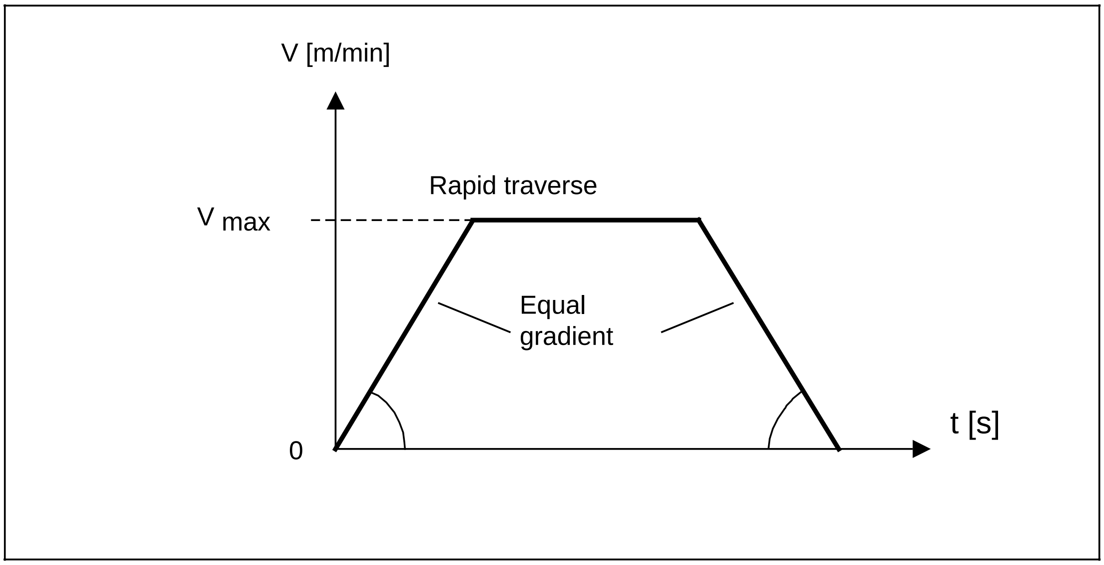
Software drift compensation is performed as follows:

1. Select **"DIAGNOSIS"** softkey
2. Press > key
3. Select **"NC MD"** softkey
4. Select MD No. 272* by means of key
5. Move cursor to desired axis
6. Key or input new value and depress key

The new compensation value is indicated in the MD.
With compensation values of more than approx. 500 VELO, the position deviation can no longer be referred to as drift since an error is involved - Alarm 160* is triggered.

It is also possible to input drift compensation manually.

276*	Acceleration		
Standard value	Lower input limit	Upper input limit	Units
50	0	4 000	10 000 units/s^2 (IS)



The axes need not be set to equal acceleration values. The control assumes the lowest acceleration value of the interpolating axes involved.

The values also apply to deceleration (braking).

Example (metric machine):

Values of 50 ... 150 (0.5 ... 1.5 m/sec^2) are customary.

280*	**Maximum speed**		
Standard value	Lower input limit	Upper input limit	Units
10 000	0	49 000 1)	1000 units/min (IS)

The input value signifies the limit speed up to which the axis may accelerate (rapid traverse limit).

Traversing is performed at this speed when rapid traverse G00 is programmed.

- Max. path speed at 5.5 ms sampling time=44.5 m/min
- Max. path speed at 4 ms sampling time=60 m/min

284*	**Reference point cutoff speed**		
Standard value	Lower input limit	Upper input limit	Units
300	0	15 000	1000 units/min (IS)

The cutoff speed is active during approach to the reference point as soon as the reducing cam is reached, i.e. the "*Deceleration" signal is active.

The feedrate override switch is not taken into account, except in the first position (0%).

Guide value:

A reasonable upper limit is 1 m/min, but values between 200 and 500 mm/min are better, depending on the servo gain factor.

288*	**Jog speed**		
Standard value	Lower input limit	Upper input limit	Units
2 000	0	49 000 1)	1000 units/min (IS)

The input value applies to travel in JOG mode with 100% feedrate override.

1) depending on NC MD 155 and the specified input, position control and display resolutions.

292*	Rapid JOG	

Standard value	Lower input limit	Upper input limit	Units
10 000	0	49 000 1)	1000 units/min (IS)

The input value applies to travel in JOG mode with the rapid traverse override key actuated and with the rapid traverse override switch in the 100% position. This value is not used for programmed rapid traverse G00. Programmed rapid traverse G00 is specified by the maximum speed MD 280*.

Guide value:

Slightly less than rapid traverse G00 to make allowance for the operator's response time.

296*	Reference point approach speed	

Standard value	Lower input limit	Upper input limit	Units
10 000	0	49 000 1)	1000 units/min (IS)

If the direction key leading to the reference point (selectable using MD 564* bit 0) is depressed in ”Reference point approach” mode, the axis accelerates to the reference point approach speed.

Exception:

Axis already at deceleration cam and automatic reference point approach selected (see Section 10.3).

300*	Incremental feedrate	

Standard value	Lower input limit	Upper input limit	Units
500	0	49 000 1)	1000 units/min (IS)

The input feed rate is only active when traversing in INC mode.

1) depending on NC MD 155 and the specified input, position control and display resolutions.

304*	Interpolation parameters		
Standard value	Lower input limit	Upper input limit	Units
0	0	3	-

In the case of circular movements (G2/G3) and thread cutting (G33), the individual axes must be assigned an interpolation parameter.

0 = No interpolation parameter
1 = Interpolation parameter I
2 = Interpolation parameter J
3 = Interpolation parameter K

Standard values:

810M/820M:	X axis	 I	(3040)
	Y axis	.... J	(3041)
	Z axis	.... K	(3042)
810T/820T:	X axis	 I	(3040)
	Z axis	.... K	(3041)

However, a possibility with special machines is, for instance, that the 5th axis has to assume the function of the X axis and must therefore operate with interpolation parameter I for circular movements.

In this case: MD 3040 = 0
MD 3044 = 1

316*	Pointer compensation +		
Standard value	Lower input limit	Upper input limit	Units
0	0	249	MD-Offset

The NC activates leadscrew error compensation after reaching the reference point. Consequently, the NC must be informed by means of MD 316* as to which of the 1000 possible compensation points represents the reference point for the axis in question (see Section 10.17).

320*	**Pointer compensation -**		
Standard value	Lower input limit	Upper input limit	Units
0	0	249	MD-Offset

If leadscrew error compensation is **direction-dependent**, the compensation curves are separate for positive and negative traversing movements. Consequently, 2 compensation indicators (MD 316* for ”+” and MD 320* for ”-”) are also required. The value refers to the compensation point corresponding to the reference point (see also Section 10).

324*	**Distance between two leadscrew error compensation points**		
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	units (MS)

The distance between two grid elements in leadscrew error compensation is based on the following:

- Permissible tolerance band
- Maximum gradient of the sum check error characteristic of the spindle/measuring system
- Maximum number of compensation points

(See also Section 10)

328*	**Leadscrew error compensation value**		
Standard value	Lower input limit	Upper input limit	Units
0	0	100	units (MS)

The compensation value depends on the permissible tolerance band for the axis position. The value for the tolerance band or a slightly smaller value is input to make use of the full bandwidth for each compensation (see also Section 10).

332*	Contour monitor tolerance band		
Standard value	Lower input limit	Upper input limit	Units
1 000	0	32 000	-

The following error is proportional to the speed after acceleration or deceleration (i.e. in the steady state), so no fluctuations in the following error must develop during travel at constant speed as this would result in contour deviations.

However, minor fluctuations in the following error triggering control processes are allowed.

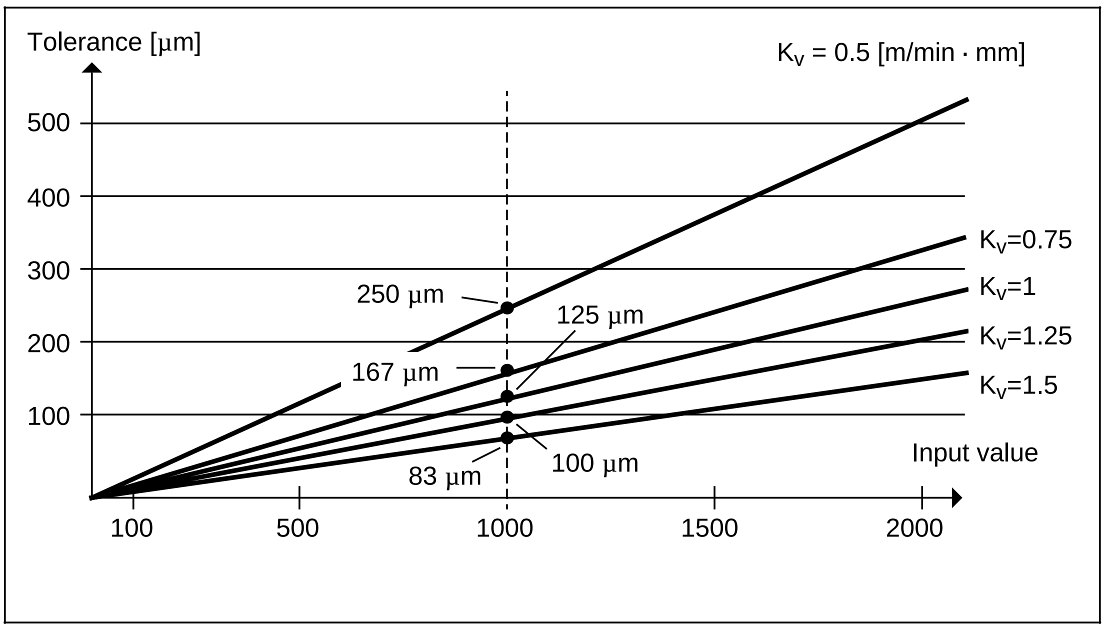
Entering a tolerance band is intended to prevent false tripping of the contour control due to slight reciprocation due to operational control processes.

The standard value of 1000 corresponds to a tolerance band of 125 µm when K_V=1

Equation for determining tolerance band:

$$\text{Tolerance in } [\mu m] = \frac{\text{MD 332*} \cdot 125}{K_v \text{ factor} \cdot 1000}$$

Example: $$\text{Tolerance} = \frac{1000 \cdot 125}{1 \cdot 1000} = 125\ \mu m$$



Calculating tolerance from the individual tolerance band curves

336*	Contour threshold speed		
Standard value	Lower input limit	Upper input limit	Units
0	1	24 000	1 000 units/min (MS)

The speed above which the contour monitor is to be active is input. Even when 0 is entered, no contour monitor is active with the axis at rest. In this case the zero-speed control checks for illegal axis movements (Alarm 112*).

340*	Tool change position		
Standard value	Lower input limit	Upper input limit	Units
0	- 99 999 999	+99 999 999	units (IS)

In the tool change cycle, the maximum retract position at which collision-free tool changing is possible is calculated by the NC from the tool and workpiece data. This MD makes it possible to specify a maximum retract position, e.g. to protect machine parts located behind (see also MD 248* Tool reference value).

344*	Rotary axis modulo value for leadscrew error compensation		
Standard value	Lower input limit	Upper input limit	Units
360 000	- 92 160 000	+92 160 000	units (MS)

This machine data is only applicable in conjunction with leadscrew error compensation. The error curve for each rotary axis must repeat cyclically. The error cycle is stored axially in this machine data.

Illegal values are not monitored and may result in uncontrolled system execution (see Section 10.17).

348*	Software Exe		
Standard value	Lower input limit	Upper input limit	Units
1	1	128 (16 000)	-

The MD is only effective for HMS, as EXE software.
The value in MD serves to multiply the encoder pulses.

Value 1 is treated internally as 1
Value 128 is treated internally as 128

The following inputs are permissible:

1, 2, 4, 8, 16, 32, 64, 128.

With inputs other than these rounding takes place to the next permissible value.

If an SPC module is used with distance-coded encoders and hardware EXE, the EXE factor must be entered in the MD 348*.

352*	Second servo gain factor (Kv factor) for thread cutting (G33)		
Standard value	Lower input limit	Upper input limit	Units
0	0	10 000	0.01 s^{-1}

For thread cutting G33 a second servo gain factor can be input in MD 352*. If this factor is for G33 = 0, the servo gain factor is taken from MD 252. If G33 is selected the servo gain factor from MD 352* becomes active.

Both servo gain factors (MD 252* and MD 352*) are **not** additive!

Weighting: 1666 corresponds to servo gain 1

360*	Symm. time constant (T_{sym})		
Standard value	Lower input limit	Upper input limit	Units
-	see Sect. 10.8.2	see Sect. 10.8.2	0.1 ms

See description in Section 10.8.2.

364*	Pulses for variable increment weighting		
Standard value	Lower input limit	Upper input limit	Units
1	1	65 000	1/2 · units (MS)

368*	Travel for variable increment weighting		
Standard value	Lower input limit	Upper input limit	Units
1	1	65 000	1/2 · units (MS)

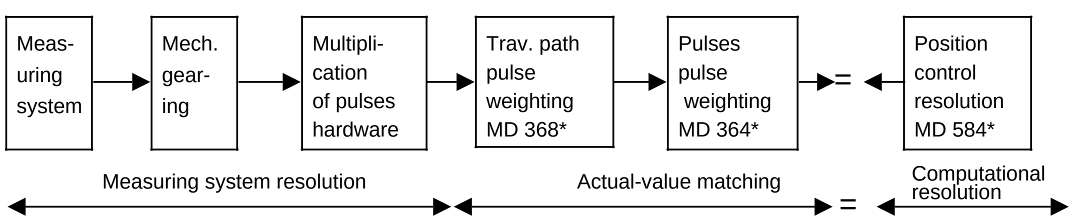
Variable pulse weighting (MD 364*, MD 368*)

In order to generate a correctly closed position control loop, the pulses from the digital measuring system and the position control accuracy of the control must be matched to one another.

In order to determine machine data 364* and 368*, the pulse number for the encoder and appropriate traversing path must be known at the machine.

The value of the traversing path is to be entered in MD 368* (in units of position control resolution). The pulse number of the encoder for this traverse path times all subsequent multiplications (EXE, measuring-system gearing, hardware 4-fold multiplication on measuring-circuit module) is to be entered in MD 364*, provided the machine data values do not exceed 65 000. In this case, both values must be divided by a common multiple.

Block diagram of position control parameters:



Determination of possible position control parameters (MD 364* and MD 368*):

1. Determination of measuring system resolution ”m”

1.1. The ROD sensor is mounted directly on the ball screw.

$$\text{MD 368*} = \frac{l}{b} \qquad \text{MD 364*} = p \cdot 4$$

1.2. The ROD sensor is mounted on the motor with gearing between the motor and ball screw.

$$\text{MD 368*} = \frac{l \cdot r}{b} \qquad \text{MD 364*} = p \cdot 4$$

1.3. A linear scale with EXE is used.

$$\text{MD 368*} = \frac{g}{b} \qquad \text{MD 364*} = f \cdot 4$$

1.4. A rotary axis is used.

$$\text{MD 368*} = \frac{l}{b} \qquad \text{MD 364*} = p \cdot f \cdot 4$$

Symbol	Machine data	Significance
b	584*, Bit 2, 1, 0	Position control resolution of control
l		Pitch of ball screw
p		Number of pulses of ROD sensor per revolution
r		Mechanical gearing between motor and ROD sensor (if fitted)
g		Period spacing on a linear scale
f		EXE multiplier
MD 364*	364*	Pulses for variable pulse weighting
MD 368*	368*	Traversing path for variable pulse weighting

2. Examples illustrating calculation of MD 364* and MD 368*:

2.1. The ROD sensor is mounted directly on the ball screw
l=10 mm
p=2500 pulses per revolution
b=1/2*10^{-3} mm

$$\text{MD 368*} = \frac{l}{b} = \frac{10\text{ mm}}{1/2 \cdot 10^{-3}\text{ mm}} = \underline{20000} \qquad \text{MD 364*} = p \cdot 4 = 2500 \cdot 4 = \underline{10000}$$

2.2. The ROD sensor is mounted on the motor with gearing between the motor and ball screw

l=0.2 inch
p=1000 pulses per revolution
r=1:2 (2 revolutions of motor=1 revolution of ball screw)
b=1/2*10^{-4} inch

$$\text{MD 368*} = \frac{l \cdot r}{b} = \frac{0.2\text{ inch} \cdot 1/2}{1/2 \cdot 10^{-4}\text{ inch}} = \underline{2000} \qquad \text{MD 364*} = p \cdot 4 = 1000 \cdot 4 = \underline{4000}$$

2.3. Same values as in example 2.2 but b=1/2*10^{-3} mm

$$\text{MD 368*} = \frac{l \cdot r}{b} = \frac{0.2\text{inch} \cdot 25.4\text{ mm/inch} \cdot 1/2}{1/2 \cdot 10^{-3}\text{ mm}} = \underline{5080} \qquad \text{MD 364*} = p \cdot 4 = 1000 \cdot 4 = \underline{4000}$$

2.4. A linear scale with EXE is used

g=0.02 mm
f=10
b=1/2*10^{-3} mm

$$\text{MD 368*} = \frac{g}{b} = \frac{0.02\text{ mm}}{1/2 \cdot 10^{-3}\text{ mm}} = \underline{40} \qquad \text{MD 364*} = f \cdot 4 = 10 \cdot 4 = \underline{40}$$

2.5. A rotary axis is used

p=18000 pulses per revolution
f=5
b=1/2*10^{-3} degrees

$$\text{MD 368*} = \frac{l}{b} = \frac{360\text{ deg}}{1/2 \cdot 10^{-3}\text{ deg}} = \underline{720000\ !!} \qquad \begin{array}{l}\text{MD 364*} = p \cdot f \cdot 4 \\ \text{MD 364*} = 18000 \cdot 5 \cdot 4 = \underline{360000\ !!}\end{array}$$

In view of the fact that the values exceed 65 000, both values must be divided by a common factor (e.g. factor=100).

MD 368*= 7200 MD 364*= 3600

2.6. Determination of the variable pulse weighting by measurement

Requirement: MD 364* = 1
MD 368* = 1

The axis is now to be traversed by a particular value (e.g.10 mm) in JOG INC or MDA mode.

The distance actually traversed is measured with a dial gauge (e.g.19.98 mm).

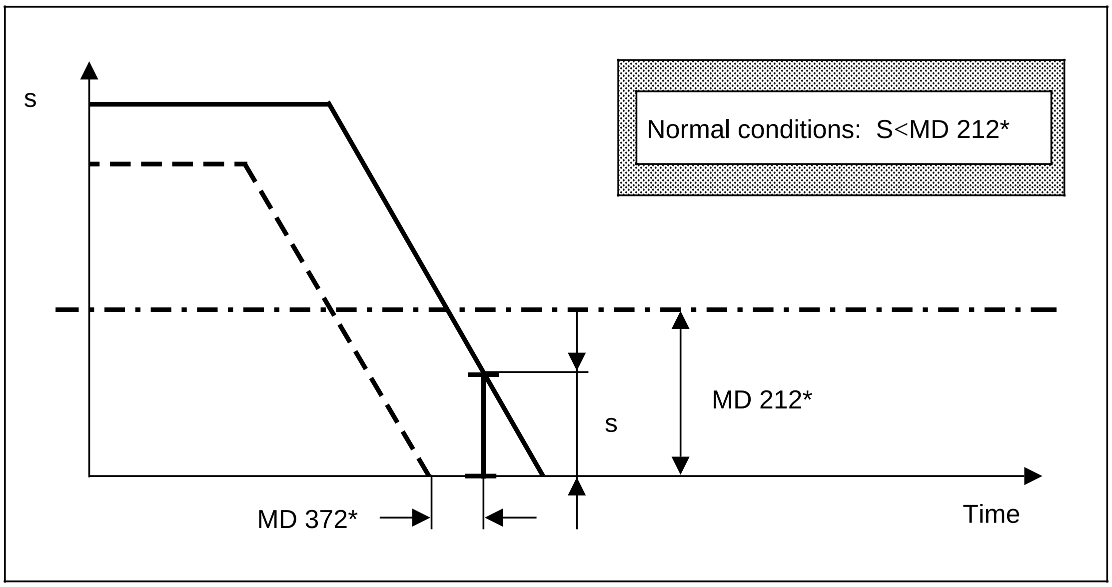
As every measurement is normally subject to error the traversed path must be rounded off, e.g. 20.000 mm.

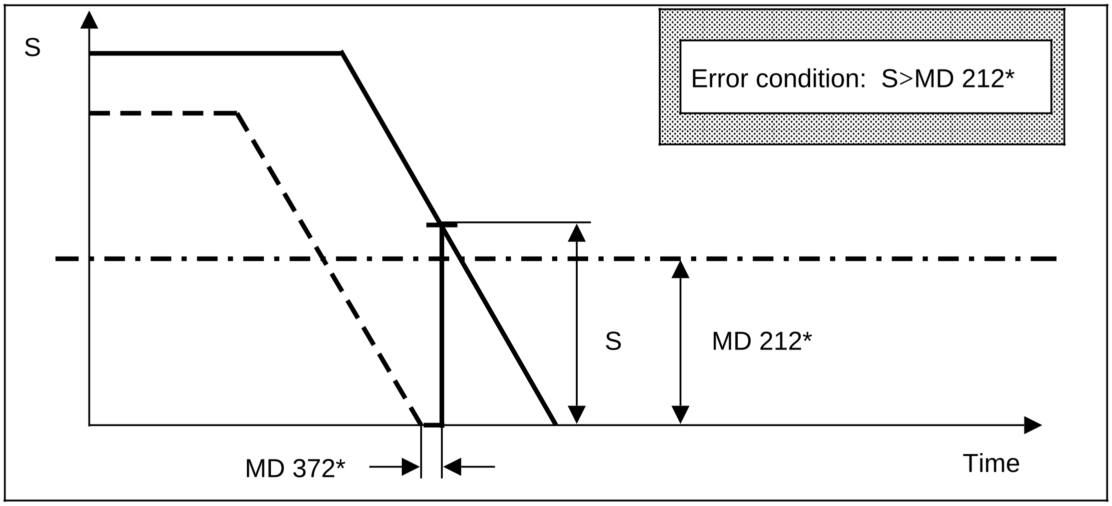
MD 364* =	distance to be traversed	10000 µm = 10000
MD 368* =	distance actually traversed	20000 µm = 20000

372*	Delay zero-speed control		
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	ms

During positioning (digital zero) the clamping tolerance NC MD 212* is activated after the time entered in this data has elapsed. The delay selected should be such that the maximum following error can be suppressed. If this is not the case, alarm 112* is activated.

If NC MD 372*=0, the value from NC MD 156 ”Cutoff delay servo enable” is used as delay for zero-speed control.





S .... Following error (contouring error)

376*	Prelimit switch		
Standard value	Lower input limit	Upper input limit	Units
20 000	0	99 999 999	units (MS)

Prelimit switch before the software (SW) limit switch: (1st or 2nd SW limit switch). It is necessary to enter here the advance distance at which the braking process is to be initiated should the momentary velocity be greater than that stored in MD No. 1. It is thus possible to ensure that the position of the software limit switch is exceeded by only a small amount in circular interpolation.

On travelling past the prelimit switch, alarm 2034 (reduction on SW prelimit switch) is issued unless travel was in rapid traverse.

Recommended value:

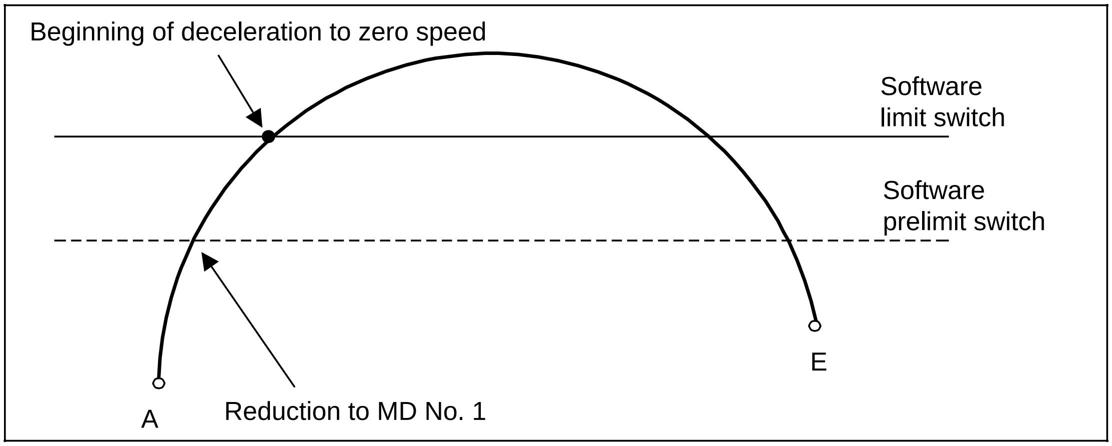
Somewhat greater than the braking path from rapid traverse to NC MD No. 1.

A travel movement whose endpoint is behind the SW limit switch is not performed at all.

Linear interpolation: If the endpoint is behind the SW limit switch, the block is not executed and alarm 2065 is issued (programmed position behind SW limit switch).

Circular interpolation:

a) If the endpoint is behind the SW limit switch, the block is not executed and alarm 2065 appears.
b) If the endpoint is not behind the SW limit switch, but if the movements continues beyond the SW limit switch, refer to the following diagram.



Schematic diagram for automatic velocity control at the prelimit switch

The region between SW prelimit switch and SW limit switch can be used fully without contouring error if the contouring velocity is less than the velocity in MD 1 (velocity behind prelimit switch).

380*	**2nd K_V factor for G36**		
Standard value	Lower input limit	Upper input limit	Units
0	0	10 000	$0.01s^{-1}$

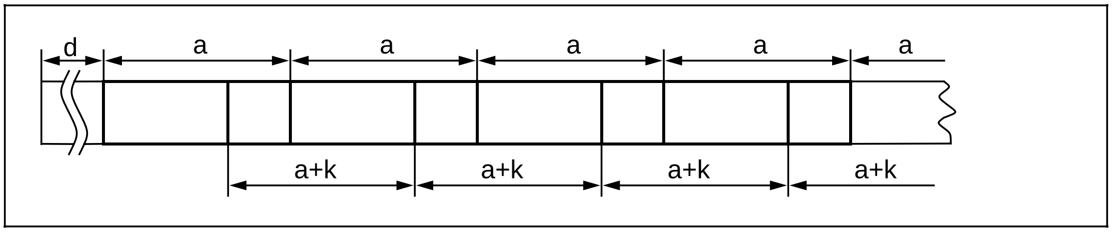
A second K_V factor can be entered in MD 380* for thread cutting G36. If this factor for G36 is equal to 0, the K_V factor of MD 252* is taken. This means that if G36 has been selected, the K_V factor from MD 380* becomes active (if ><0).
The two K_V factors (MD 252* and MD 380*) do not act additively.
Value: 1666 corresponds to K_V 1.

384*	Graduation of linear scale		
Standard value	Lower input limit	Upper input limit	Units
0	0	32 000	units (MS)

This machine data must be used for the function ”Distance-coded reference marks”. Consequently, the following MDs have no significance:
MD 244*, MD 296*, MD 560* bit 6

The MD 564* bit 0 remains valid.

The graduation unit of the linear scale varies with different manufacturers. Its thousand-fold value corresponds to the basic spacing of the reference marks.



K = Graduation period of the linear scale
a = 1000 · K
d = Distance of first reference mark from the beginning of the measuring length

The ”Distance-coded reference marks” function requires the use of the measuring circuit module with RPC in the hardware.

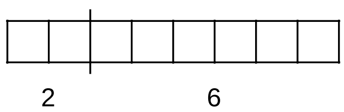
388*	**Axis specific weighting factor**		
Standard value	Lower input limit	Upper input limit	Units
0	0	99 999 999	-

With the axis specific weighting factor it is possible to program a specific axis value but to traverse a prepared axis value.

Conversion formula:

prepared axis value = programmed axis value·weighting factor

Eight-digit input of the MD:



2 Integer places

6 Decimal places

For every input except zero the first two places are interpreted as integer places and the remaining six as decimal places.

e.g. input 12345678 gives a weighting factor of
12.345678

If the setpoint exceeds the internal format because of the weighting factor or too high a speed, the Alarm 2031 ”Weighting factor too large” is set and NC start and machining are disabled.

Standard value: 0 (Weighting factor 1)

392*	**Absolute offset**		
Standard value	Lower input limit	Upper input limit	Units
0	- 99 999 999	99 999 999	units (MS)

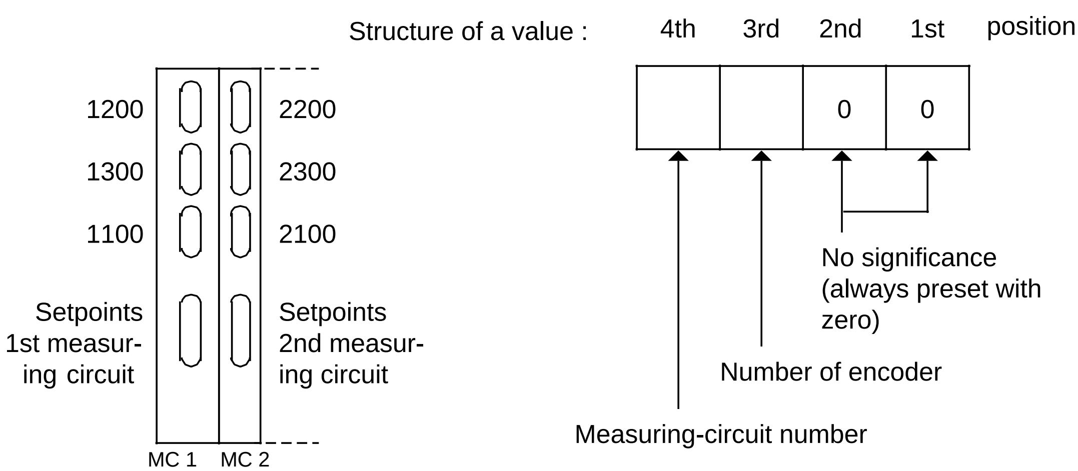
Absolute offset = reference point value - displayed absolute position

The absolute offset is used for calculating the absolute position for reference point approach of axes with distance-coded reference marks.

400*	**Spindle assignment**		
Standard value	Lower input limit	Upper input limit	Units
see possible values	0000	see possible values	-

This machine data is used to specify the measuring-circuit connector on which the actual and set (command) spindle values are connected.

- **Definition when using the SPC module**



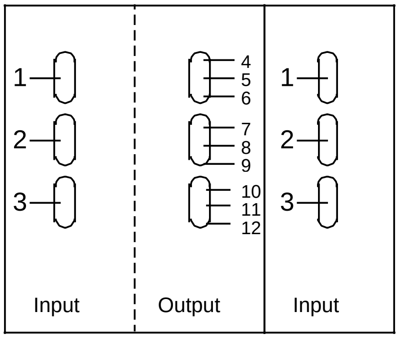
Possible values:

0000	Axis not available at machine (only permissible with MD564* bit 7=0)
1100	Measuring circuit 1, encoder 1
1200	Measuring circuit 1, encoder 2
1300	Measuring circuit 1, encoder 3
2100	Measuring circuit 2, encoder 1
2200	Measuring circuit 2, encoder 2
2300	Measuring circuit 2, encoder 3

Standard assigment: 810M GA3/820M GA3: 0
810T GA3/820T GA3: 1300

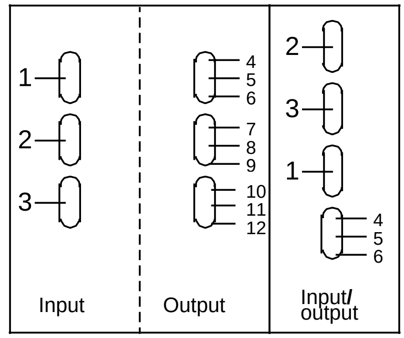
Spindle-specific values — *0 spindle 1*, *1 spindle 2*

- **Definition when using the HMS module**



1st HMS meas. circ. (-6BA..)   2nd HMS meas. circuit (-6BB..)

or



1st HMS measur-ing circuit   2nd SPC module

The measuring circuit interfaces are specified as shown below:

00 02	00 03	00 02	04 12
Number of measuring circuit module Input	Number of measuring circuit connection Input	Number of measuring circuit module Output	Number of measuring circuit connection Output

The spindle assignment is thus complete for each spindle if MD 400x has been defined for the spindles.

Example: MD4002 = 1 0 2 0 2 0 4

Measuring circuit module 1 is 6FX1 145-6BAxx (HMS),
Measuring circuit module 2 is 6FX1 121-4BAxx (SPC).

This means:
– the actual value must be read in from the 2nd connection of the HMS module,
– the setpoint must be output to the 4th connection of the SPC module.

The previous definitions of MD 200x can further be used if the module involved is a SPC module.

The NC recognizes by the number of places whether the previous or the new definition is used.

401*	**Spindle drift compensation**		
Standard value	Lower input limit	Upper input limit	Units
0	0	500	VELO

The input value must be modified in the appropriate direction until the spindle exhibits equal actual speeds in both directions of rotation.

It must be adjusted at low speeds. It is checked with reference to the indication in the basic display (for spindles with encoder) or using a rev counter.

402*	**Zero mark offset for spindle**		
Standard value	Lower input limit	Upper input limit	Units
0	0	max. 16383	approx. $\frac{360}{\text{MD 4590 x 4}}$ deg.

With this machine data you can offset the zero mark for the spindle.

When the spindle is synchronised (Power On or PLC signal Q103.3/Q 107.3 ”Resynchronise spindle”) the spindle value is not cleared but the value of MD 402* is taken over. If the MD 402* is overwritten by @ 400 in the running program the zero mark offset takes immediate effect according to the following block format:

@ 714
@ 400 K402* K=value (e.g. 4020)
@ 714.

Application example: G33 or M19
One increment of the pulse encoder is for:

- 1024 encoders, in MD 4 59* = 1024

  $$\frac{\text{340 degrees}}{\text{4 x MD 4590}} = \frac{\text{360 degrees}}{\text{4 096 pulses}} = \text{0.08789 ... degrees} \qquad \text{approx. 1/11 degrees}$$

- 2500 encoders

  $$\frac{\text{360 degrees}}{\text{10 000 pulses}} = \text{0.036 ... degrees}$$

Note:
If encoders are used with more than 16380/4 pulses, the encoder position might have to be adjusted mechanically. This is particularly important for dual track encoders.

403*... 410*	Maximum speed for 8 gears		
Standard value	Lower input limit	Upper input limit	Units
	0	16 000	rev/min

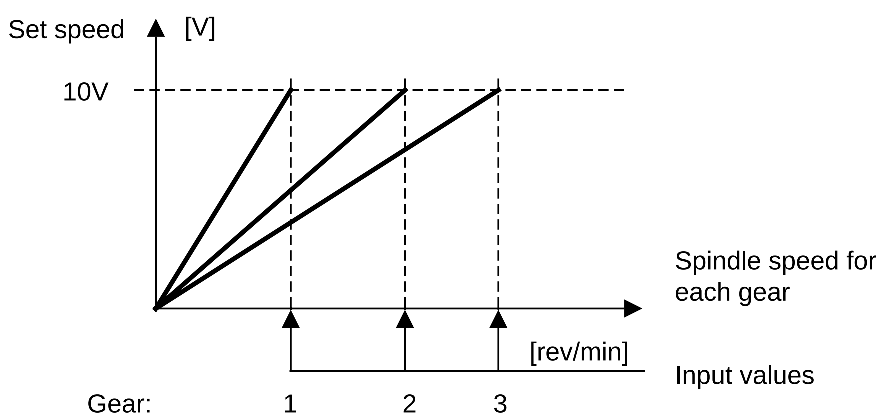
Assignment:

Gear	1	2	3	4	5	6	7	8
MD No.	403*	404*	405*	406*	407*	408*	409*	410*
Stand. value	500	1000	2000	4000	4000	4000	4000	4000

If spindle encoders have been fitted, the maximum speed depends among other things on the number of graduations on the encoder (see MD 459*).

The machine data specify the maximum spindle speed reached in the individual gears with a setpoint input of 10V.

The value 0 is entered for unavailable gears.



The maximum programmable spindle speed is 16 000!

MD 459* must be considered.

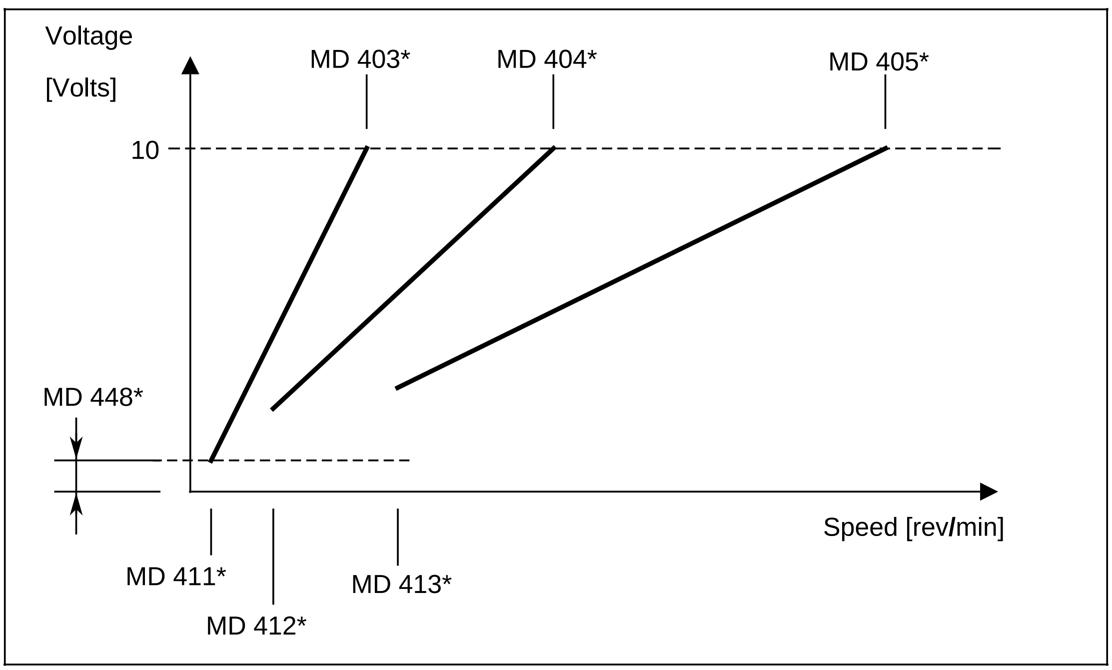
411*... 418*	Minimum speed for 8 gears		
Standard value	Lower input limit	Upper input limit	Units
	0	16 000	rev/min

When MD 5200.3 is set, the unit is 0.1·rev/min.

Assignment:

Gear	1	2	3	4	5	6	7	8
MD No.	411*	412*	413*	414*	415*	416*	417*	418*
Stand. value	50	500	1000	2000	2000	2000	2000	2000

Inputting of the minimum and maximum speeds in the respective gears means that their speed range is now defined. On the basis of the programmed spindle speed, the NC can now transmit the required gear and a request signal for a gear change to the PLC, selection in the event of overlapping speed ranges being based on the lowest operating frequency (selection via ”Automatic gear selection”, Q 101.3/Q 105.3).



If 3rd gear has been engaged, the new S value must be less than MD 413* to ensure that the NC activates a gear change.

419*... 426*	Accelerating time constant for 8 gears		
Standard value	Lower input limit	Upper input limit	Units
2 000	0	16 000	4 ms

Assignment:

Gear	1	2	3	4	5	6	7	8
MD No.	419*	420*	421*	422*	423*	424*	425*	426*

The control specifies the setpoint for acceleration in ramp form as a function of this machine data. The machine data acts as a variable ramp-function generator. It is set by measuring each time interval from zero speed to maximum speed.

This time is entered in the machine data. The standard value of 2000 corresponds to a time of 8 seconds.

427*... 434*	Cutoff speed for M19 for 8 gears		
Standard value	Lower input limit	Upper input limit	Units
100	0	2 000	rev/min

This machine data indicates the spindle speed to which reduction takes place in the event of oriented spindle stop (M19) and at which travel continues until positioning is performed via the gain on the basis of the set position control characteristic. When entering values > 2000, the NC reduces these values to 2000.

Assignment:

Gear	1	2	3	4	5	6	7	8
MD No.	427*	428*	429*	430*	431*	432*	433*	434*

(See also Section 10.1.3)

435*... 442*	Gain for M19 for 8 gears		
Standard value	Lower input limit	Upper input limit	Units
200	0	10 000	rev/min / 360 deg

Assignment:

Gear	1	2	3	4	5	6	7	8
MD No.	435*	436*	437*	438*	439*	440*	441*	442*

In the event of oriented spindle stop (M19) spindle closed-loop position control is activated. The gain is defined by means of the positioning gradient to the cutoff position. The gradient is defined as the spindle speed (in rev/min) given a positional variation of 360° (See also Section 10.1.3).

443*	Positional tolerance for M19		
Standard value	Lower input limit	Upper input limit	Units
11	0	4 096	ca. 1/11 deg

In the case of oriented spindle stop (M19) the message ”SPINDLE POSITION REACHED” is output to the PLC via input I 114.4 or I 116.4 as soon as the positional variation lies within this tolerance.
However, the spindle closed-loop position control still tries to reduce the positional variation further.

The positional tolerance is stated in pulse encoder increments. 1 increment corresponds to 360/4096 degrees. Given a pulse encoder with 1024 pulses per revolution, hardware four-fold multiplication yields 4096 pulses per revolution. Only pulse encoders with 1024 pulses (512 pulses) are permissible. The standard value of 11 corresponds to roughly 1°.

The spindle closed-loop position control remains active until:

- PLC NC signal ”Spindle enable” (Q 100.7 and Q 104.7) is cancelled.
- PLC NC signal ”Acknowledge M19” (Q 103.2 and Q 107.2) is received.
- M3 or M4 is programmed in NC part program.

(See also Section 10.1.3)

Corresponding MD: MD 520* bit 5, MD 520* bit 6

444*	Spindle speed tolerance		
Standard value	Lower input limit	Upper input limit	Units
10	0	100	%

The difference between the actual speed and set speed is determined for systems with analog spindle speed and spindle encoder.

The actual speed is measured by means of ROD sensors. Deviations outside the tolerance limit for the programmed spindle speed are signalled to the PLC via input I 114.5 or I 116.5 by cancelling the ”Spindle in set range” signal.

The tolerance (rev/min) is obtained from the entered tolerance in % with respect to the set speed.
(Set speed-tolerance) - actual speed- (set speed+tolerance)
The monitor is disabled at 100 %.

Example:

- S value: 1000 rev/min
- Tolerance in MD: 3 %

yields a permissible actual speed range of 970-1030 rev/min

445*	Maximum spindle speed tolerance		
Standard value	Lower input limit	Upper input limit	Units
100	0	100	%

In systems with analog spindle speed and spindle encoder, a deviation beyond the maximum speed plus tolerance limit activates the ”Speed limit exceeded” signal (I 114.0 or I 116.0 in PLC) and Alarm 2152. The smallest of the following maximum spindle speed limits then becomes active:

- Max. gear speed (MD 403* - 410*)
- Max. spindle speed (MD 451*)
- For G96: value in setting data (G92 S ...)
- Spindle speed limit (setting data)

The monitor is disabled at 100 %.

446*	Zero-speed tolerance		
Standard value	Lower input limit	Upper input limit	Units
10	0	16 000	0.01 %

Unit: 0.01 % of maximum gear speed

The actual speed is measured in the case of systems with analog spindle speed and spindle encoder. Undershoot of the zero speed is reported to the PLC as a ”Spindle stationary” signal (I 114.3 or I 116.3).

447*	Servo enable cancelling delay		
Standard value	Lower input limit	Upper input limit	Units
1 000	16	16 000	ms

When spindle setpoint 0 is output, servo enable for the spindle is cancelled after this delay to prevent drifting.

The delay is active for:

- Cancelling of servo enable for spindle.

448*	**Minimum motor set speed**	

Standard value	Lower input limit	Upper input limit	Units
50	0	8 192	VELO

This machine data specifies the minimum motor speed which must not be undershot, for instance, at constant cutting speed and as the turning diameter increases. In other words, as of this point the cutting speed is no longer constant but increases with the turning diameter. The MD should be set such that smooth running of the motor is always guaranteed.

Example:

MD 4480=50
50 x 1.22 mV=61 mV

449*	**Basic speed**	

Standard value	Lower input limit	Upper input limit	Units
50	0	9 999	rev/min

If the "Basic speed" signal (Q 103.5 or Q 107.5) is activated by the PLC, a spindle setpoint corresponding to this spindle speed is output, taking into account the gear just selected. Spindle override is active. (See also Interface Description Part 1)

450*	**Set reciprocation speed**		
Standard value	Lower input limit	Upper input limit	Units
50	0	8 192	VELO

The (low) spindle **motor speed** entered in this MD takes effect if the ”RECIPROCATION SPEED” signal (Q 103.6 or Q 107.6) is activated by the PLC. (See also Interface Description Part 1).

The standard value of 50 corresponds to a value of 61 mV.

Note:

Selecting a reciprocation speed does not yet result in reciprocation. This must be generated via ”Set direction of rotation clockwise/counter-clockwise” (Q 103.7 or Q 107.7).

451*	**Maximum spindle speed**		
Standard value	Lower input limit	Upper input limit	Units
4 000	0	16 000	rev/min

When MD 5200.3 is set, the unit is 0.1·rev/min.

Exceeding this maximum spindle speed by more than the tolerance limit (MD 445*) results in output of the ”Speed limit exceeded” signal and setting of Alarm 2152 (spindle speed too high). (See also MD 445*).

Note:

The maximum programmable spindle speed is 16 000.

MD 459* must be considered here.

452*	**Spindle position for external M19**		
Standard value	Lower input limit	Upper input limit	Units
0	0	3 599	0.1 degrees

If M19 is initiated from the PLC with the ”Position spindle” PLC signal (Q 103.4 or Q 107.4), the NC positions the spindle at the angle entered in MD 452*.
(See Interface Description Part 1).

459*	Number of spindle encoder pulses		
Standard value	Lower input limit	Upper input limit	Units
1 024	0	16 000	Pulses per encoder revolution

When pulse encoders other than those with 1024 pulses are used for the spindle, the corresponding number of pulses per spindle revolution is entered (without quadruplication). As a result, the screen displays are converted to the corresponding values. Any rotational feedrate that might be used also refers to this value. The accuracy for the M19 function, thread cutting and synchronisation, is not increased however.

The spindle must be resynchronised after the machine data has been changed (e. g. by POWER ON). Otherwise positioning will not be correct.

When a spindle encoder with a higher number of pulses is used, the possible maximum speed must be taken into account.

The maximum speed depends on the number of pulses and the IPO sampling interval (T_{Ipo}).

The maximum speed is normally calculated as follows:

$$n_{max} = \frac{60 \cdot n_{limit}}{N} \quad \text{e. g.} \quad \frac{60\,000\,000}{N} \quad \text{in [rev/min]}$$

for encoders with 1 MHz limit frequency.

n_{limit} = limit frequency of encoder in Hz
N = number of pulses per spindle revolution

Examples for n_{limit} = 1 MHz

N	n_{max} [min^{-1}]
1 024	(58 000) 16 000
2 500	(24 000) 16 000
5 000	12 000
9 000	6 666

The maximum speed is limited to 16 000 rev/min.

461*	Rotary axis assignment		
Standard value	Lower input limit	Upper input limit	Units
0	0	Max. number of axes available	–

Assignment of rotary axis to spindle.

Error message 8 is output when

- the limit value for the number of axes is exceeded
- or the axis does not exist or is a simulation axis
- or the spindle has been assigned no measuring circuit in MD 400* and no pulse weighting by MD 520*.

Error message 2183 is output if the axis is not defined as rotary axis.

Value 0 specifies that no rotary axis has been assigned to the spindle.

The following values are possible:

Axis 1 = 1
Axis 2 = 2
....
Axis 7 = 7
No axis= 0.

8.3 Description of the NC MD bits

The significance of the individual MD bits

always refers to the set bit.

If a bit is not set, the statement (name) is to be negated.

General bits

5000							
7	6	5	4	3	2	1	0
				Name of RADIUS and CHAMFER			

Name of radius and chamfer for:

- Contour definition
- Circular-path programming
- Polar coordinates
- Diameter ratio for cylinder milling

Coding of addresses (axis names)

7 … 4	3 … 0	Bit No.
	Address name	

X = 0000
Y = 0001
Z = 0010
A = 0011
B = 0100
C = 0101
U = 0110
V = 0111
W = 1000
Q = 1001
E = 1010

e. g. 0000 0010 = Z
0001 1001 = Q1

5001							
7	6	5	4	3	2	1	0
				Name of ANGLE and POSITION			

Name of angle and position for contour definition.

The position of the tool nose centre (SINUMERIK 810 GA3T/820 GA3T) which, with other systems, was also specified in address A, is defined in TO parameter P1 (type) for tool offsets in the case of the SINUMERIK 810 GA3T/820 GA3T.

Coding of addresses (axis names):

7 … 4	3 … 0	Bit No.

Adrdess name

X = 0000
Y = 0001
Z = 0010
A = 0011
B = 0100
C = 0101
U = 0110
V = 0111
W = 1000
Q = 1001
E = 1010

e. g. 0000 0010 = Z
0001 1001 = Q1

5002							
7	6	5	4	3	2	1	0
Input resolution			Reset position G70				Position control resolution Inch

Bit 4 Reset position for input and display resolution
Bit 4 = 0 Reset position is G 71 (metric)
Bit 4 = 1 Reset position is G 70 (inch)
Bit 0 Bit 0 = 0 Measuring system metric
Bit 0 = 1 Measuring system inch

If the metric measuring system has been set, it is possible to write a part program in the inch system.

Bit 7, 6, 5 The input resolution defines the incremental weighting for the dimensional inputs and the smallest possible traverse path. The possible values, the combinations of input, position control and display resolution are described in Section 10.7 ”Difference input, position control and display resolutions”. Changes become effective only after POWER ON.

Please note the changes compared with SW2.

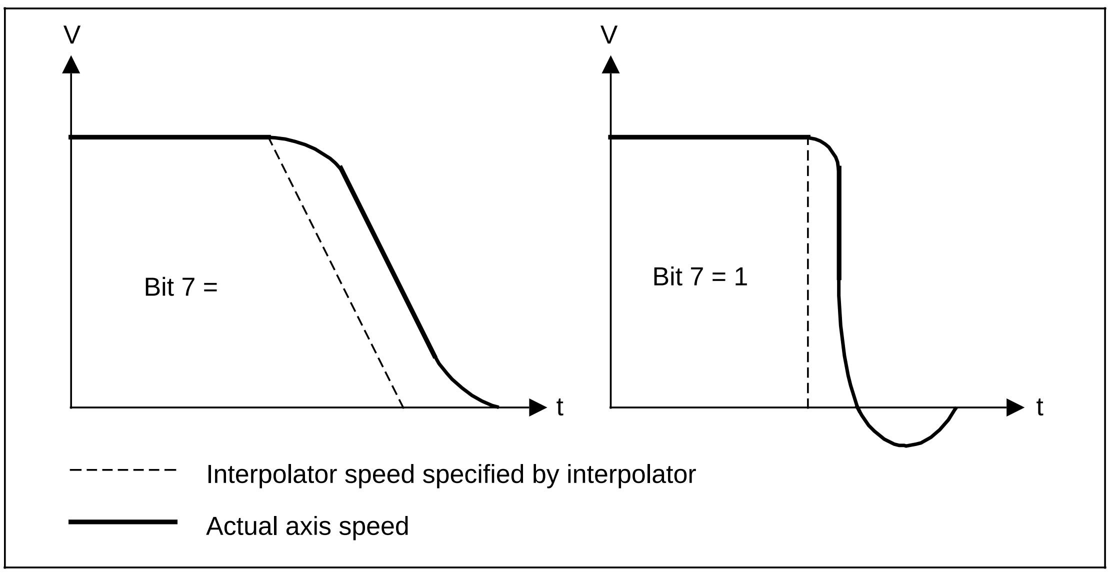
The position control resolutions must be specified in NC MD 584*.

General bits

5003							
7	6	5	4	3	2	1	0
No deceleration at limit switch	JOG working area limitation active	Interpol. parameters I, J, K depending on G90 / 91	Polar coordinate angle depend. G90 / 91	Do not clear PRESET OFFSET at Power On	Auxiliary function output before travel	Simulation with G59	Simulation with G58

Bit 7 No deceleration on reaching the software limit switch.
With the bit set, braking is not performed over the acceleration/deceleration characteristic, only the following error being suppressed. The limit switch is not overrun to this extent (see also MD 376*).

In the case of interpolating axes, all axes are shut down if the range limit of an axis has been reached. However, stopping without contour violation is only guaranteed if MD 5003 bit 7 (no deceleration at limit switch) has been set, i.e. during braking via the acceleration ramp.



Bit 6 In addition to the software limit switches, the working area may be limited in JOG mode and the machine better safeguarded against unintentional travel. Since, however, working area limitation is a software limitation, proper operation can only be ensured after reference point approach.

Bit 5 When this bit is set, interpolation parameters I, J, K (not contour definition interpolation parameters) are programmed as a function of G90/G91. If bit 5 = 0, the interpolation parameters are output as incremental dimensions.

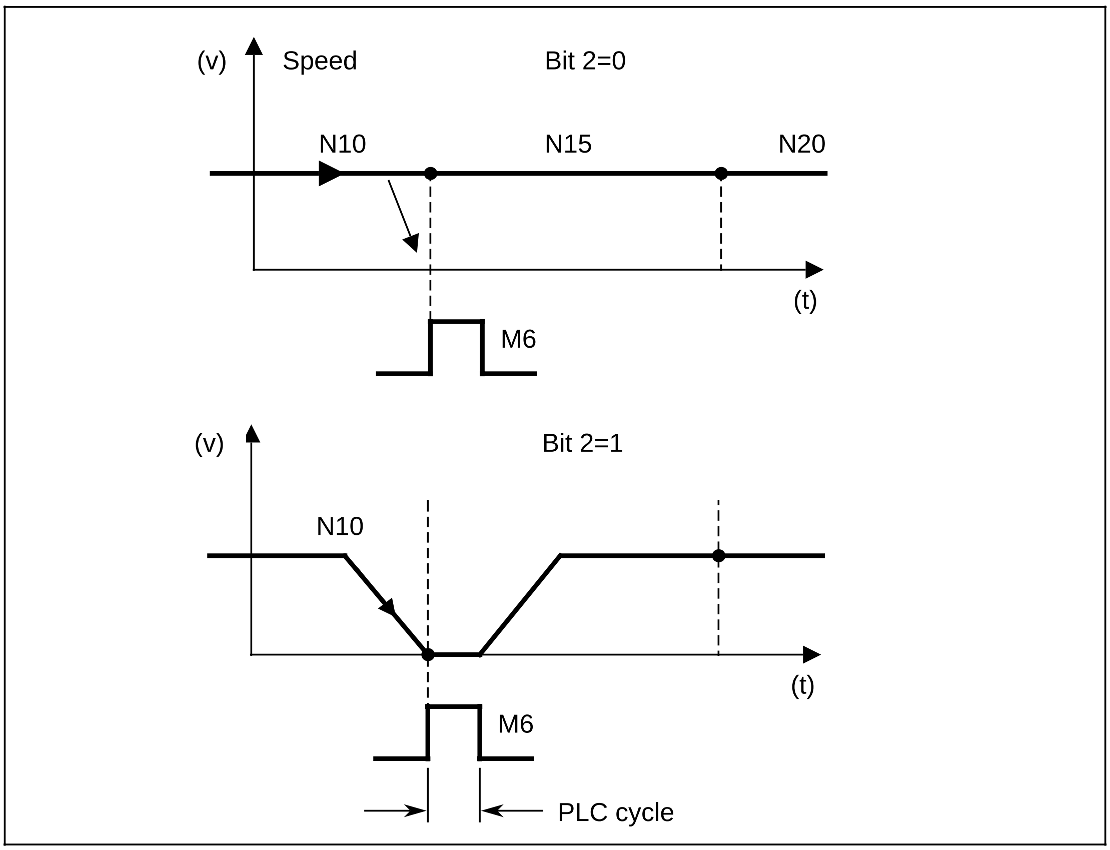
Bit 4 When this bit is set, the polar coordinate angle is programmed as a function of G90/91. If bit 4 = 0, the angle is specified as an absolute dimension.

Bit 3 The old PRESET OFFSET is assumed automatically once more in the actual-value display after switching on.

Bit 2 Bit 2=0 Auxiliary function output during axis movement
Bit 2=1 Auxiliary function output before travel

If an auxiliary function (M, S, T, H) is also programmed in a traversing block, the following two conditions may occur through bit 2:

Example: N 10 G1 X10 F1000 LF
N 15 M6 X100 LF
N 20 ...



Bit 1, 0 Work is always performed in graphic simulation without tool offsets, zero offsets and software limit switches.

G58/G59 (prog. ZO) can nevertheless be included in the calculations (mould making cycles) as a function of these two MD.

5004							
7	6	5	4	3	2	1	0
			Own rapid traverse override	NC start without ref. point	3rd	2nd	1st
					handwheel available		

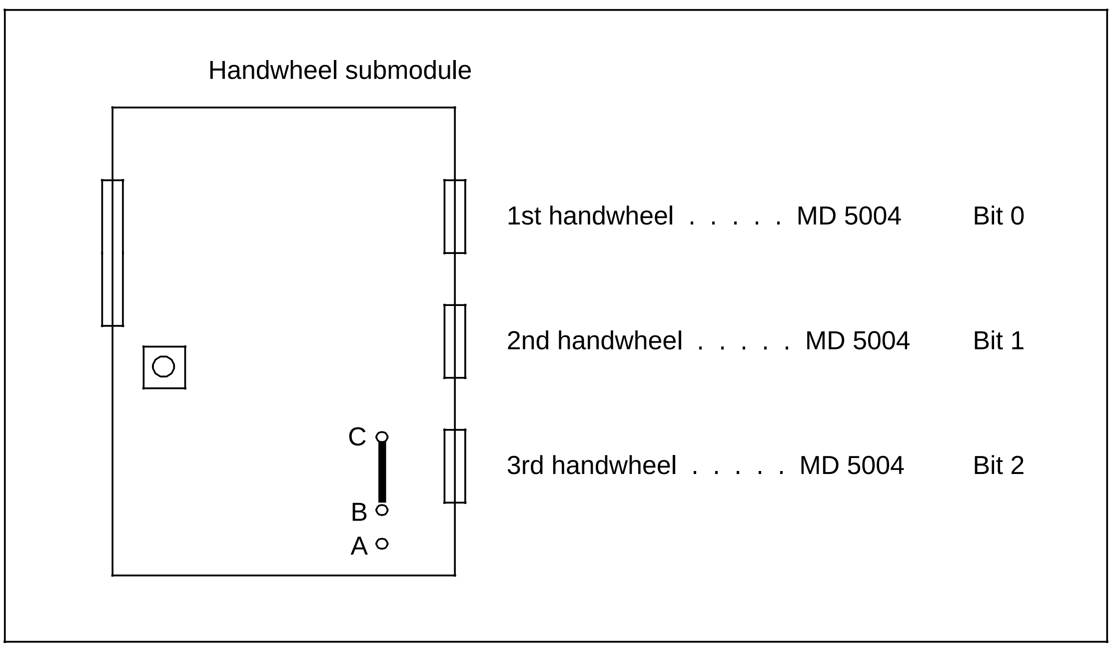
Bit 4 The rapid traverse override switch usually featuring 4 positions (MD 147 - 150) must be transferred the user program. It is only active with G0.

Bit 3 The ”NC START” PLC signal (Q 87.0 and Q 96.0) initiates program start even if the reference points have not been approached, i.e. the feed axes are not synchronized with the machine position. The software limit switches, however, do not operate in jog mode without reference point approach.

In automatic mode, however, they do still operate.

In order to ensure correct workpiece machining, synchronization must then be otherwise performed, e.g. using the scratch method.

Bit 2,
1, 0 The number and locations of the available handwheels are specified.



5005							
7	6	5	4	3	2	1	0
Keyswitch group (configurable)			ZO additive	ZO protected R parameters	KEYSWITCH active with: TO wear	TO geometry	Reserved

Bits 7,6,5 With the 810 GA3/820 GA3 configurable, the keyswitch can be used to block any changes to values in the case of displays with data input (INPUT, EDIT, CLEAR). The programming workstation (WS 800) is used to specify assignment of the keyswitch group to the value concerned (e.g. R parameters, input buffer parameters, ...). A maximum of 3 keyswitch groups is available for configuring purposes.

Bits 4, 3,2, 1 By setting the appropriate bits for the following data, it is not possible to modify without the keyswitch.

Bit 4 : Zero offsets (G54 - G57) additive
Bit 3: Zero offset (G54 - G57)
Protected R parameter ranges (cycle SD) See also MD 16,17
Bit 2: TO wear (P5 - P7)
Bit 1: TO geometry (P2 - P4)

5006							
7	6	5	4	3	2	1	0
Channel-specific program infl.		KEYSWITCH active with: MD and overall reset	TEACH-IN PLAY-BACK	PP manual input	Dry run	DRF	Over-store

Bit 7 When this bit is set the signals for program influencing (softkeys), delete block, DEC single block, dry run feedrate and M01 active are only transmitted on a channel-specific basis to IB 104 or 110.

Bits 5-0 Setting the corresponding MD bits makes it possible to disable input by means of the keyswitch for the following operations:

Bit 5: Selection of NC MD menu tree, select installation overall reset mode
Bit 4: Teach in, playback
Bit 3: Input of part programs via operator panel (not MDI)
This also inhibits handling of part programs (COPY, MOVE, RENAME, DELETE, REORG).
Bit 2: Selection of dry run feedrate (DRY RUN)
Bit 1: Traversing via handwheel in automatic mode (DRF)
Bit 0: Transferring (overstoring) of H, S, M, T functions.

5007							
7	6	5	4	3	2	1	0
TO over diameter	TO wear not to be included	Mixed programs G90/91 in block	Enable simul-taneous simula-tion	Basic tool dimen-sion active	No output of M17	G53 as for @ 706	Length compen-sation also with non-prog axes

Bit 7 Bit 7 = 0 Cutter parameter P4 (tool parameter P1 = 20) is defined as radius (P4).

Bit 7 = 1 Cutter parameter P4 is defined as diameter. (The cutting edge radius of the turning tool is always defined as radius.)

Bit 6 All tool wear data can be declared invalid (P5-P7). A value can then be added to or subtracted from the tool geometry data using the ”EDIT” key.

Bit 5 Mixed programming of G90 and G91 in one block.

Bit 4 When simultaneous simulation (simulation during machining) is enabled there is a risk of the machine not performing properly since machining and simulation are not completely separate in the NC.
When this bit is set, the customer must be made aware of the particular dangers (e.g. programming of G58 X, Y, Z also affects the machine coordinate system during simulation).

Bit 3 When this bit is set, tool offset parameters P8 and P9 are active.

Bit 2 Bit 2= 0 SUBROUTINE END (M17) is issued to the PLC as an M function.

Bit 2= 1 Subroutine end is only active internally in the NC.

Bit 1 Bit 1= 0 With G53 all zero offsets (G54-G59 + ext. ZO) are cancelled.

Bit 1= 1 With G53 all zero offsets (G54-G59 + ext. ZO), DRF and PRESET are cancelled. Tool offset (TO) is not cancelled.

Bit 0 Scan length compensation even if axes are not programmed:
If the change in tool length compensation (e.g. cancelled by means of D0) yields a traversing path in an axis which is not programmed in this block, traversing is nevertheless performed. If the bit is not set, traversing is only performed when the axis has been programmed.

Example of programming for 810T/820T:

```
.
N5 G17 G0 X0 Y0 LF
N10 G01 F200 D3 X10 LF
N20 Z30 LF
.                    D3 ....   Type 9 length 20
```

With the bit set, the length compensation (X axis) is already moved in block N10 to position X20 (does not apply for SINUMERIK 810M/820 M).

When the bit is not set, the X axis is not moved to position X50 until block N20 (X30+length compensation).

5008							
7	6	5	4	3	2	1	0
	REPOS in inching operation	Setting up in inching operation	TO type 0= type 20 (for M only)	External customer machine control panel	Internal handwheel selection	External machine control panel	Internal machine control panel or operator panel

Bit 6 Bit 6 = 0 In REPOS mode, return to the contour is initiated with brief depressing of the appropriate direction key.

Bit 6 = 1 Return to the contour is only performed while the corresponding direction key is being depressed.

Bit 5 As for bit 6 but for setting up modes (REF. POINT, INC, . .)

Bit 4 Bit 4 = 1 If no tool type is defined, the NC interprets this as type 20, i.e. end mill (length 1=3rd axis of the plane, radius acts in the 1st and 2nd axes of the plane). No alarm is set. Only with M software!

Bit 4 = 0 If no tool type is defined, Alarm 2060 ”Programming error tool. ZO” is set.

Bit 3 The bit must be set if use is made of coded selector switches on the external machine control panel which are not identical with the switch on the standard machine control panel
(PLC MD 2002 bit 3 must be set to 0).

When bit 3=1 another Gray code is evaluated for axis selection switches 1 and 2 (only with M controls).

Bit 3=0	Bit 3=1
Siemens machine control panel	Customer machine control panel

Position	E	D	C	B	A
X axis	0	0	0	1	1
Y axis	0	0	0	1	0
Z axis	0	0	1	1	0
4th axis	0	1	1	0	0
5th axis	0	1	1	1	1
6th axis	0	1	0	1	0
7th axis	0	1	0	0	1

Position	E	D	C	B	A
X axis	0	0	0	0	1
Y axis	0	0	0	1	1
Z axis	0	0	0	1	0
4th axis	0	0	1	1	0
5th axis	0	0	1	1	1
6th axis	0	0	1	0	1
7th axis	0	0	1	0	0

The input area for the customer machine control panel is the same as for the Siemens machine control panel (IB0-3). In this way the user can use different axis selector switches.

Bit 2 In order to enable flexible assignment of the handwheels to the axes with one or more handwheels, this assignment can be performed by means of SD (axial). The handwheels are enabled by means of softkeys (”handwheel” softkey in INC (1-10000)).

Bit 1 The bit must be set when use is made of an external machine control panel.

Bit 0 The bit must be set when use is made of the internal machine control panel or the integrated operator panel.

5009							
7	6	5	4	3	2	1	0
				Blank optimization string output		M machine control panel (for SINUMERIK T only)	

Bit 3 If bit 3 equals one, unnecessary blanks are not output on the screen as it is cleared when it is changed anyway. This makes picture construction about twice as fast. If it is not possible to suppress the output of blanks in configured pictures, this bit must be set to zero.

Bit 1 For SINUMERIK 810T/820T

When bit 1 is 0, an external T machine control panel can be connected.
When bit 1 is 1, an external M machine control panel can be connected.

5011							
7	6	5	4	3	2	1	0
	Diameter functions for transverse axis						
@ read/ load over diameter	Actual-value display over dia.	Diameter prog. with G91	Diameter prog. G90 tool offset wear over diameter	Tool offset geom. over diameter	INC, hand-wheel DRF over diameter	Zero offset over diameter	

Bit 7 @ reading or writing of transverse axis values over radius or diameter depending on MD bits (MD 5011, bits 1-6).

Bit 6 Actual value display of transverse axes over diameter.

Bit 5 Diameter programming with G91 for transverse axes.

Bit 4 Diameter programming with G90 for transverse axes.
Turning tools (tool types 1 - 9): input of wear over diameter for transverse axis.

Bit 3 Turning tools (tool types 1 - 9): input of geometry over diameter for transverse axis.

Bit 2 INC. (incremental dimension), handwheel increments, DRF offset.
Input and display over diameter for transverse axes.

Bit 1 Programmable (G58- G59) and settable (G54 - G57) zero offsets; display and programming over diameter.

External zero offset, PRESET offset, distance to go, JOG offset (REPOS): display over diameter.

When bit 4 or bit 3 is set, movements and displays do not conform if MD bit "Transverse axis" (572*.1)=0.

Treatment of transverse axis values in @ commands

In view of the fact that accuracy of only 1 unit of input resolution can be achieved when reading and writing values via the @ command, but the radius values in diameter I/O have 1/2 units of input resolution, reading and writing of diameter values must also be possible. A new machine data bit is introduced for this purpose. If this bit is set to 1, then the machine data bits stated in MD 5011 for facing axes are also requested in the @ commands.

MD 5011, bit 7= 0 @ reading/writing of transverse axis values over radius

MD 5011, bit 7= 1 @ reading/writing of transverse axis values over radius or diameter-dependent on MD bits

The table below applies:

@ command	Data for transverse axis	Machine data 5011
320/420/423	Tool offset TO - types 1-9 Tool length (P2) Tool wear (P5)	 Bit 3 Bit 4
330/430	Settable ZO (G54 . . G57, coarse/fine)	Bit 1
431	Settable ZO additive (G54 . . G57)	Bit 1
331/432	Programmable ZO (G58/G59)	Bit 1
333/434	DRF offset	Bit 2
334/435	PRESET offset	Bit 1
360	Workpiece-related actual value	Bit 6
361	Actual control value	Bit 6
440	Programmable axis position	Bit 4

Command @336 always yields a radius value.

Commands @3FF and @4FF process the data in the above table as radius or diameter values even if MD 5011 bit 7=1.

5012							
7	6	5	4	3	2	1	0
Input PLC alarm texts					Disable overwrite of MD with @ com-mand		

Bit 7 PLC alarm texts for Alarms Nos. 6000-6063 and 7000-7063 can be read in via the RS232C (V.24) interface when bit 7=1.

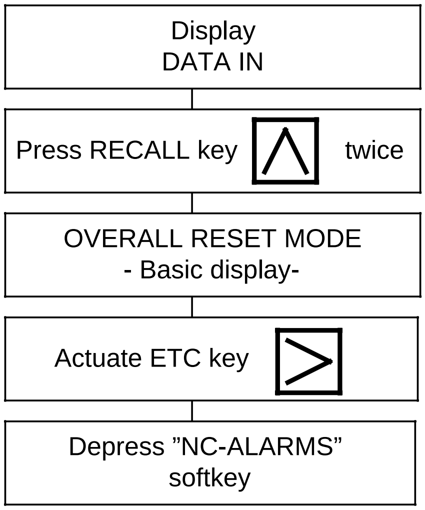
Sequence:
- Set MD 5012 bit 7
- Select ”overall reset installation” mode
- Actuate ”NC DATA” softkey
- Actuate ”FORMAT. AL-TEXT” softkey
- Press RECALL key
- Read in alarm texts

Format:

% PCA LF
N6000 = TEXT (max. 36 characters) LF
.
.
N6031 = TEXT (max. 36 characters) LF
N7000 = TEXT (max. 36 characters) LF
.
N7031 = TEXT (max. 36 characters) LF
M2 LF

The blank ahead of the ”**=**” must be written. When the program header ”% PCA LF” is being read in, all old alarm texts are erased. The alarm texts are stored in RAM on the CPU.

”**(**” may be written as the separator instead of ”**=**”.



– During alarm text transmission ”DIO” and ”PCA” are displayed.
If ”DIO” and ”PCA” disappear **before** transmission has been completed, a RS232C (V.24) alarm is activated. Since the ”PLC Stop” alarm appears on top of the RS232C alarm, the RS232C alarm can only be displayed in the NC alarm display.

RS232C alarms can only be acknowledged by means of the ”STOP” softkey.

Bit 2 MD 5012, bit 2 disables writing of machine data via @40.

5012. Bit 2=1	Writing disabled
5012. Bit 2=0	Writing enabled (Standard MD)

Writing disabled acts on the following @ command:
@400, @401, @402, @406, @407, @408, @409, @40a, @40b.

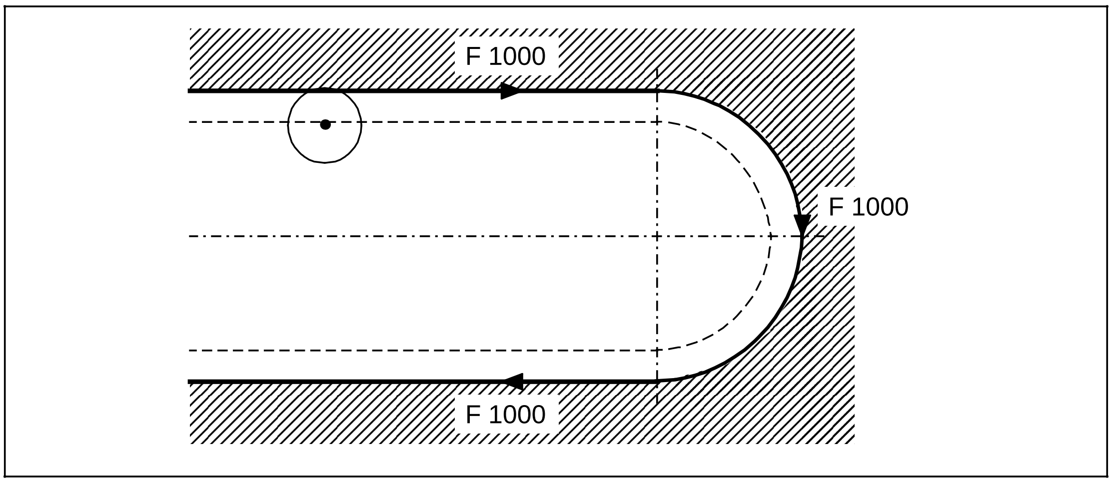
If MD 5012. bit 2=1 and @40 is programmed,
Alarm 3004 ”CL800 error” appears.

5013							
7	6	5	4	3	2	1	0
Circle radius prog.			Feed not contour-related		Add address extension for M and S	Tapping without encoder	G63 without feedrate reduction

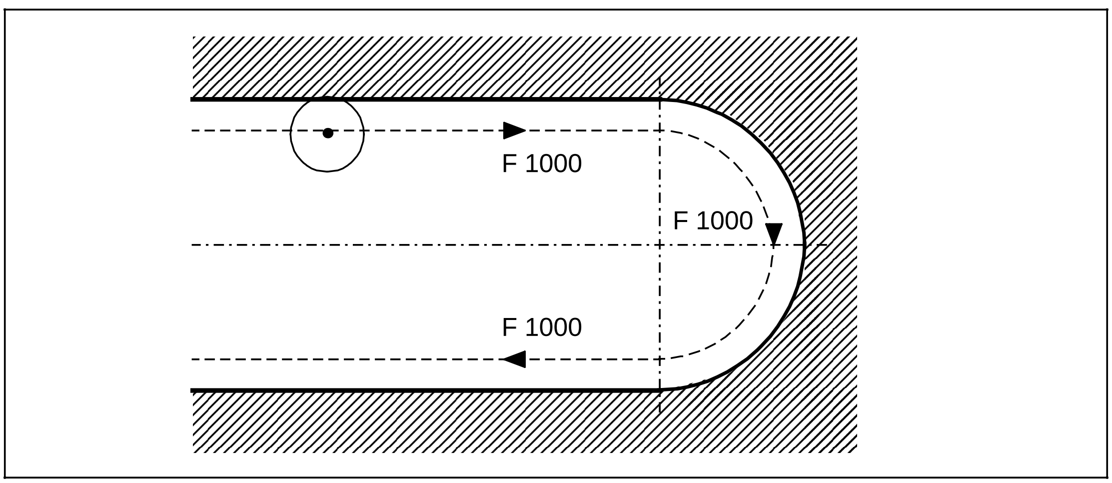
Bit 7 With the bit set, a circle can be programmed by specifying the radius and/or angle.

Bit 4 The programmed feedrate refers to the cutter centre path (tool nose radius centre path).

Example: M version
Bit 4=0 (feedrate at contour constant)



Bit 4 = 1 (feedrate of cutter constant)



Bit 2 If there is more than one spindle, it might be necessary to work with address extension for M and S.

Bit 2 = 0: Add no address extension to M and S. In this case, the spindle function always refers to the leading spindle. The address extension is not output at the NC/PLC interface.

Bit 2 = 1: Add address extension to M and S. With M03, M04, M05, M19 and S the leading spindle number is added as address extension if no spindle number is programmed. The added address extension is output at the NC/PLC interface.

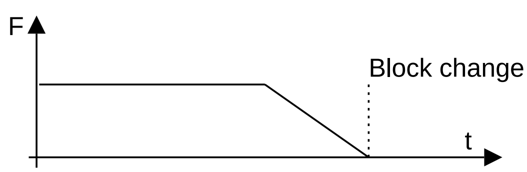
Bit 1 During tapping using standard cycle L84 (G84), the spindle can be used with or without a pulse encoder (ROD encoder).

Bit 1= 0: The spindle has a pulse encoder (512 or 1024 pulses). In the tapping cycle L84, G33 (pitch I, J, K in mm/rev) is therefore used.

Bit 1= 1: The spindle has no pulse encoder.
In the tapping cycle L84, G63 (F in mm/min) is therefore used, and the programmer must define the feedrate and the speed to achieve the correct thread pitch. Slight errors are compensated for by the compensating chuck.

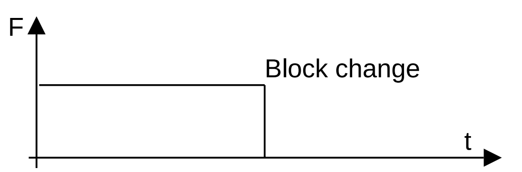
Bit 0 Bit 0 = 0 G63 with feedrate reduction: the control behaves as for G09/G60. The feedrate is reduced at the end of the block.

– Exact stop G09/G60
– Programmable feedrate limitation



Bit 0 = 1 G63 with feedrate reduction: the feedrate is not reduced at the end of the block.

• End-of-block reduction is not possible



5014							
7	6	5	4	3	2	1	0
TNRC/ CRC		Cycles (ref. cond.)					

Bit 7 TNRC/CRC: 810 GA3T/820 GA3T: Activating of tool nose radius compensation (G41/G42)
810 GA3M/820 GA3M: Activating of cutter radius compensation (G41/G42)

Bit 5 Cycles: This bit activates reference conditioning (software submodule), which is of vital importance for cycle processing.
The bit must always be set to "1".

5015							
7	6	5	4	3	2	1	0
	User memory submodule			ASCII string without erase colour	Graphics		

Bit 6 Bit 6 = 1 Select UMS.
POWER ON must be carried out once more after the bit has been set.
The control must be started with the UMS (format user memory!).
Bit 6 = 0 Cancel UMS.
No cycles can be activated any more. No changed system menu can be activated.

Bit 3 Bit 3 = 1 ASCII string without erase colour. Picture can be constructed faster.

Bit 2 Bit 2 = 1 Graphics is possible (e. g. contour definitions)
Required for SIMULATION, contour definitions etc.

Bit 2 = 0 Only text is displayed.

The bit is set as the default value.

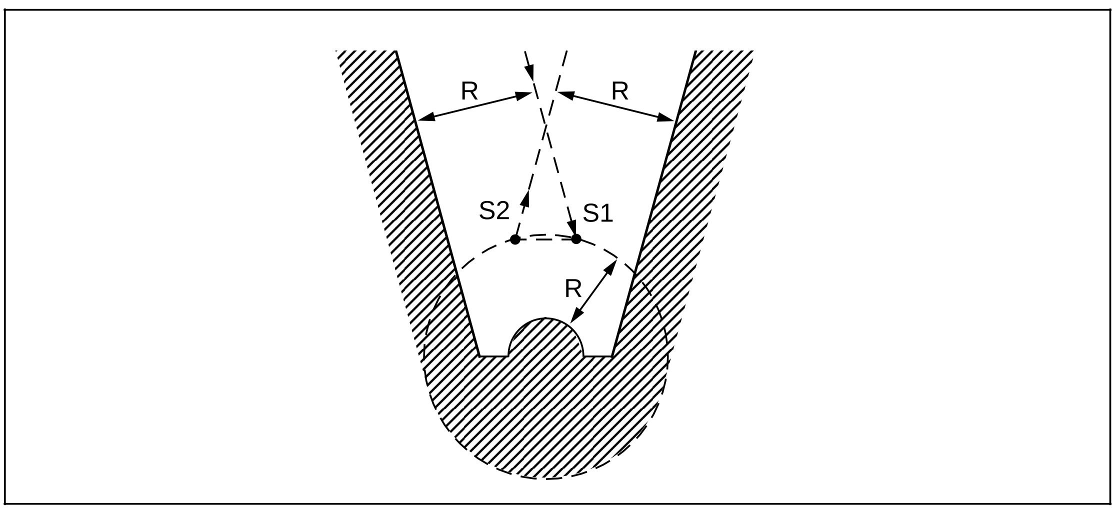
5016							
7	6	5	4	3	2	1	0
	3964 R with BTR						

Bit 6 When the BTR option has been selected this machine data is used to select secured reading in (procedure 3964 R) or unsecured reading in (Xon/Xoff).

Bit 6 = 1 selects secured data transfer
Bit 6 = 0 selects unsecured data transfer

5018							
7	6	5	4	3	2	1	0
Switch off CRC contour monitoring							

Bit 7 With circle-circle and circle-line transitions overlapping of the CRC vectors may occur. The control no longer performs any circular compensating movement but adopts NC stop status and activates the alarm message 3012 ”Contour violation with CRC/TNRC”. The program can be continued by pressing the cancel key and the NC start key. If contour violations are considered to be acceptable, monitoring can be switched off via MD 5018 bit 7.



5061	Transformation type						
7	6	5	4	3	2	1	0
							Transmit

Bit 0 = 0 No transformation

Bit 0 = 1 Transmit if the Transmit option is available. MD 5061 to MD 5066 have no significance without the Transmit option.

5062							
7	6	5	4	3	2	1	0
Axis name of 1st fictitious axis							

1111 1111 must be entered if the axis is not involved in transformation.

The axis name must be entered if the axis is involved in transformation (see description for NC-MD 568*).

5063							
7	6	5	4	3	2	1	0
Axis name of 2nd fictitious axis							

As for NC MD 5062.

5064							
7	6	5	4	3	2	1	0
	Axis name of infeed axis						

As for NC MD 5062.

5065							
7	6	5	4	3	2	1	0
	Axis name of 1st real axis						

As for NC MD 5062.

5066							
7	6	5	4	3	2	1	0
	Axis name of 2nd real axis						

As for NC MD 5062.

Spindle-specific bits

520*							
7	6	5	4	3	2	1	0
Spindle override active for thread cutting	No M19 abort on RESET	M19 with axis movement		Speed in 0.1 rev/min	Pulse encoder available	Actual value sign change	

Bit 7 **Caution:**

When the bit is set, a change in the following error (due to modified spindle speed) produces a fault at the thread

Bit 6 Bit 6=1 M19 is active until spindle enable (Q 100.7 or Q 104.7) is cancelled (MD 5210, bit 4=0).
Bit 6=1 M19 is active until the "Acknowledge M19" signal (Q 103.2 or Q 107.2) is issued (MD 5210, bit 4=1).
RESET or M02/M30 does not interrupt M19
Bit 6=0 M19 is interrupted immediately on RESET or M02/M30

Bit 5 Bit 5=1 Signifies no waiting for positioning of the main spindle; instead, the next program block (possibly with axis movement) is executed after output of M19 to the PLC (simultaneous positioning of axes and spindle) (see also Section 10.2)

Bit 3 The programmed S word is given the dimension (0.1 rev/min), i.e. the speed range lies between 0.1 (rev/min) and 1600 (rev/min). The MD also acts with G96 on the cutting speed S.

Note:

Spindle setpoint/actual value comparison must be performed as follows.
NC program M3 S1000 (i.e. 100 rev/min)
Spindle actual value S100! (display)
Otherwise the interface signal "Spindle in setpoint range" is not received!

Bit 2 Must be set if a function calling for a spindle pulse generator is requested.
- G36 (Tapping with interpolation)
- G33 (Thread cutting)
- M19 (Oriented spindle stop)
- G95 (Feedrate per revolution)
- G96 (Constant cutting speed)

The pulse encoder is assigned by means of MD 400* and MD 459*.

Bit 1 Resetting the bit produces a software change of sign in the spindle encoder pulses.

* Spindle-specific bits 0 spindle 1
1 spindle 2

521*							
7	6	5	4	3	2	1	0
Spindle available	Separate spindle reset	Spindle speed transfer after acknowl.	Acknow-ledge M19 by PLC or M03/M04	No improved M19 position-ing	C axis mode retained after RESET	Setpoint sign change	Spindle and axis on one measuring circuit

Bit 7 Spindle available
Up to 5 axes and one spindle can be connected. All spindle MDs are first activated with this bit 7 (including spindle override).

This bit is only active after POWER ON.

Bit 6 With this bit set the Reset key on the machine control panel and M02/M30 has no effect on the spindle. The separate spindle reset (Q 101.5 or Q 105.5) is active when the control is in the reset state.

Bit 5 Spindle speed transfer only after acknowledgement by the PLC.
With the bit set and with ”Automatic gear selection” (Q 101.3 or Q 105.3) the ”Switch gear” signal (I 115.7 or I 117.7) must be reset by the PLC user program. Only then is the new set speed output.

Bit 4 Bit 4=1 Acknowledge M19 by means of

1. PLC signal ”Acknowledge M19” (Q 103.2 or Q 107.2)
2. M03/ M04 from NC program (if MD 5200 bit 5 = 1)

Bit 4=0 M19 is acknowledged by cancelling spindle enable (Q 100.7 or Q 104.7).

Bit 3 M19 Gain characteristic
When positioning the spindle via M19 the 810/820 responds to changes as small as a 4096th against a 1024th of a measuring pulse. This enhancement is active if this bit is not set, NC MD 521*.3=0.

Bit 2 If this bit is equal to 1, a selected C axis mode remains active even after RESET. If this bit equals 0, then it changes to spindle mode after RESET and if M02/M30 has been programmed it changes to spindle mode for a specific channel.

Bit 1 Resetting the bit produces a change of sign for the setpoint (command value).
With bit 1=0 a positive set speed is output to the main spindle for M03.
The setpoint may also be reversed from the PLC by means of the ”Invert spindle speed” interface signal (Q 101.4 or Q 105.4).

Bit 0 With bit 0 equals 1 and MD 461* equals 1...7 :

Changeover between rotary axis mode and spindle mode is effected with M functions that are defined in the MD 260 for selecting axis mode and MD 261 for selecting spindle mode.
The measuring circuits in the MD 200* and MD400* can be assigned as required. This means that the actual value input and the setpoint output can be the same or separate.

Example: Axis 1 actual value on measuring circuit 1
Spindle 1 actual value on measuring circuit 2
Axis 1 setpoint on measuring circuit 5
Spindle 1 setpoint on measuring circuit 6
and hence MD2000 = 1010105 and
MD4000 = 1020106

With bit 0 equals 1 and MD 461* equals 0

A common actual value input and a common setpoint output is used for the rotary axis and spindle or (second case) a common actual value input and a separate setpoint output. Changeover between spindle mode and rotary axis mode is effected here with the PLC signal ”parking axis”.

Signal ”parking axis” = 1 — Spindle mode on. The PLC signal ”ref. point reached” for the axis is deleted.

Signal ”parking axis” =0 — Rotary axis mode on. The PLC signal ”spindle synchronous” is deleted.

Channel-specific bits

540*[1)]							
7	6	5	4	3	2	1	0
Transmit not cancelled with Reset	Feedrate in $^{m}/_{min}$	0	0		Analog spindle speed with T version		Auxiliary functions to PLC

Bit 7 Bit 7 = 0 In the JOG and INC modes, fictitious axes can only be traversed if an Automatic program has previously been interrupted during a Transmit operation. On changeover to the operating modes ”Reference point approach”, MDI or Preset, the transformation is cancelled as with actuation of the Reset key.

Bit 7 = 1 Transmit remains active in RESET status. Transmit is cancelled when the operating mode is changed after reference point approach. Transmit is also cancelled by the NC program command G130.

1) channel-specific bits

Bit 6 Bit 6=0 Reset (default)
Feedrate in [mm/min]

Bit 6=1 Feedrate in [m/min]

This machine data is active in all metric input systems.

The maximum axial speeds specified in the machine data remain unaffected by the new machine data. Display of the feedrate value depends on the machine data. The feedrate value of F external and dry run feed is not converted to [m/min].

Bits 5,4 These bits must always be set to 0.

Bit 2 This bit must be 1 if a setpoint is to be output to the spindle.

Bit 0 Bit 0=0 inhibits output of auxiliary functions to the PLC.

546*1)							
7	6	5	4	3	2	1	0
	Auxiliary functions not skipped during block search but output immediately						No aux. function output with block search
0	H	0	T	S	M	0	

Bits 6, 4,3,2 The auxiliary functions specified are not skipped with block search but are output immediately (may result in several switching functions being triggered in rapid succession via the PLC).

Bit 0 If the bit is set all auxiliary functions are skipped with block search.

Examples of possible bit patterns:

	Bit pattern for bits ... 6	5	4	3	2	0	Significance
Example a)	No relevance					**1**	**No output of auxiliary functions**
Example b)	**0**	**0**	**0**	**0**	**0**	**0**	**All auxiliary functions are skipped and the last aux. function output after NC start**
Example c)	**1**	**0**	**1**	**1**	**1**	**0**	**All auxiliary functions are output during block search**
Example d)	**0**	**0**	**0**	**0**	**1**	**0**	**M functions are output during block search, skip H, T, S functions**
Example e)	**1**	**0**	**0**	**0**	**0**	**0**	**H functions are output during block search, skip T, S, M functions**
• • etc.			• • etc.				

1) *channel-specific bits*

548*1)							
7	6	5	4	3	2	1	0
Name of abscissa (horizontal axis) (1st axis of cube) (same code as for axis definition)							

550*1)							
7	6	5	4	3	2	1	0
Name of ordinate (perpendicular axis) (2nd axis of cube) (same code as for axis definition)							

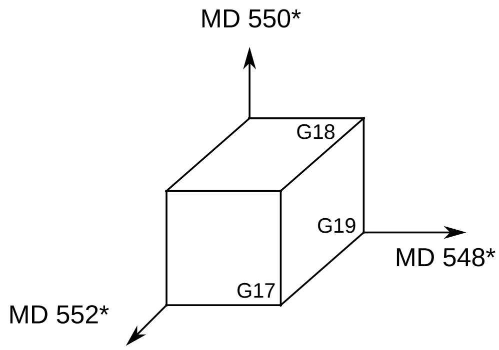
552*1)							
7	6	5	4	3	2	1	0
Name of applicate (vertical axis) (3rd axis of cube) (same code as for axis definition)							

These MDs specify the axes defining the space (planes) with G17, G18 and G19. The machine data must be specified with valid axis names. This basic setting (Reset position) is specified in the channel-specific MD 110*. The axis names must be the same as defined in MD568*.

The following correlation exists between the MDs 548*, 550*, 552*and the G functions G17, G18 and G19.

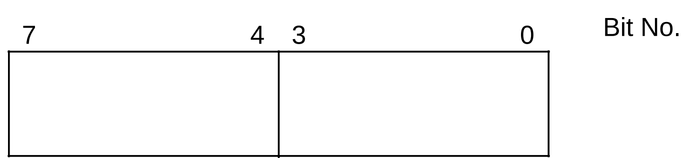
1) *channel-specific bits*

G17	G18	G19
548*	552*	550*
550*	548*	552*
552*	550*	548*



Note: The meaning of the MDs has changed and is no longer compatible with the previous meaning.

Coding of addresses (axis names)



Extended addresses (address number)

blank =0000
1 =0001
2 =0010
3 =0011
4 =0100
5 =0101
6 =0110
7 =0111
8 =1000
9 =1001

Address name

X = 0000
Y = 0001
Z = 0010
A = 0011
B = 0100
C = 0101
U = 0110
V = 0111
W = 1000
Q = 1001
E =1010

e. g. 0000 0010 = Z
0001 1001 = Q1
0010 1001 = Q2

554*1)							
7	6	5	4	3	2	1	0
Number of the axis with constant cutting rate							

Number of the axis with constant cutting rate G96
Machine date 554* determines which axis is to be the reference axis for the function of constant cutting rate G96 S... .

MD 554* 0000 0000 $\hat{=}$ axis 1
0000 0001 $\hat{=}$ axis 2
0000 0010 $\hat{=}$ axis 3
0000 0011 $\hat{=}$ axis 4
0000 0100 $\hat{=}$ axis 5
0000 0101 $\hat{=}$ axis 6
0000 0110 $\hat{=}$ axis 7

558*1)							
7	6	5	4	3	2	1	0
						No alarm 3021 when D No. changes	Add axes after block search

Bit 1 Bit 1 = 0 When D No. changes, a check is made for contour violation. An alarm is possible with machine stop.

Bit 1 = 1 When D No. changes (D No. > 0), no check is made for contour violation. Alarm message 3021 is suppressed. The machine does not remain in the stop state.

Caution: Contour violation possible.

1) *channel-specific bits*

Bit 0 Bit 0 = 0 Axes are not added after block search and NC start. Only the axes programmed in the target block are traversed.

Bit 0 = 1 Axes are added after block search.

Programming example (current axis pos.: X33, Y93, Z-25):

```
.
N5   G0   X0    Y0     Z0
N10       X100  Y2100  Z100
N15  T1   D1
N20  M50  X200
.
```

If a search to block N20 is made, the other axes are traversed to the position of block N10 after NC start and M50 is output.

Position:	before block search:	X33	Y93	Z-25
Position:	after block search	X33	Y93	Z-25
Position:	after NC start			
Position:	or processing of N20:	X200	Y2100	Z100

The block traversed after block search is executed with linear interpolation (possible risk of collision).

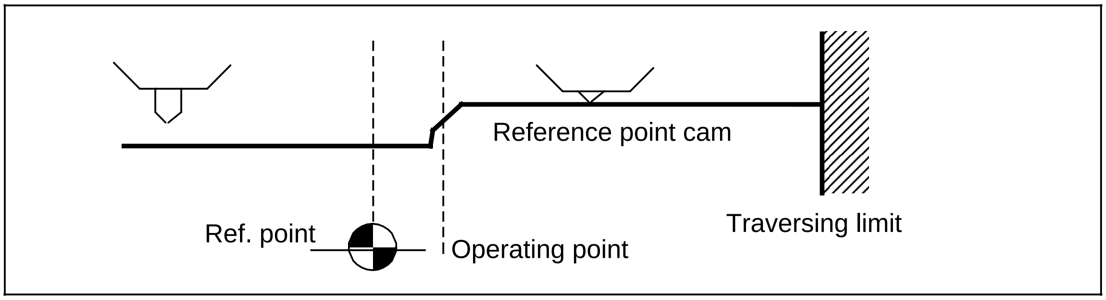
Axis-specific bits

560*							
7	6	5	4	3	2	1	0
Display actual value modulo 360°	Automatic ref. point approach	Software limit switch working field lim. active	No start inhibit for ref. point	Rounding for rotary axes	Rounding to whole/half degrees		No meas-uring circuit monitoring

Bit 7 For rotary axes only! The actual-value display jumps from 359.999 to 0 degrees after one revolution of the rotary axis.

Bit 6 Bit 6=0 If the axis is between the reference point cam and EMERGENCY STOP after the NC has been switched on, the axis moves to EMERGENCY STOP during approach to the reference point since the NC cannot detect from the ”*Deceleration” interface signal whether the axis is ahead of or behind the reference point cam.

Bit 6=1 The NC can detect precisely the direction of the reference point from the ”*Deceleration” interface signal since the reference point cam extends as far as the traversing limit (see Section 10.3).



Bit 5 Bit 5=0 Causes the software limit switches (software limit switches 1 and 2) and working area limitations to be overrun without response.

Bit 4 Bit 4=1 A program can be started with NC START without reference point approach in this axis. Certain axes can therefore be excepted from reference point approach using this bit.

Bit 3, 2 May only be used for rotary axes.

Active for rotary axes and linear axes.

In jog mode, rounding to whole or half degrees is performed (positioning) as a function of bit 2 so that the rotary table can be correctly lowered into the ring gear. In automatic mode, Alarm 2064 (rounding for rotary axis incorrectly programmed) is activated for unrounded increments.

Bit 3=1 means rounding activated
Bit 2=1 means rounding to whole degree

Bit 0 Bit 0=1 Alarm 132* is disabled (control loop hardware). The measuring circuits are no longer monitored for cable breaks.

564*							
7	6	5	4	3	2	1	0
Axis exists	Fictitious axis	Rotary axis			Actual value sign change	Setpoint sign change	Ref. point in negative direction

Bit 7 The set bit causes the axis to appear on the screen and the position controller and measuring-circuit monitor to be activated or to be fictitious (for Transmit).

Note:

The position controller and measuring-circuit monitor are only active after Power On, although the axis address is displayed immediately on the screen.

Bit 6 Bit 6 = 0 Axis is real or not available.
Bit 6 = 1 Axis is fictitious and is part of the transformation group.

Bit 5 If an axis is declared a rotary axis, the following functions assume a different significance:

- Programming in degrees
- The axis-specific machine data is specified in degrees
- Endless rotation possible.

Combinations of possible rotary axis functions:

NC MD 564*. 5 Rotary axis	NC MD 572*. 2 Modulo program	NC MD 560*. 7 Actual modulo	
0	0	0	Linear axis
1	0	0	Resolution 10^{-3}, range overflow correction, absolute display, @ function modulo, absolute data channel transfer, programming as for linear axis. Endlessly turning not possible, because actual value display overflows at 99 999.999 degrees.
1	0	1	Resolution 10^{-3}, range overflow correction, modulo display, @ function modulo, modulo data channel transfer, programming as for linear axis
1	1	1	Resolution 10^{-3}, range overflow correction, modulo programming (display @ function data channel transfer) modulo
1	1	0	Use not allowed
0	0	1	Use not allowed
0	1	1	
0	1	0	Use not allowed

Bit 2 The signs of the measuring-system pulses may be interchanged by resetting the bit. Necessary when the axis traverses uncontrollably at maximum speed or if Alarm 112* (zero-speed control) is set during commissioning with standard MD.

Bit 1 Resetting of the bit produces a change in the polarity of the speed controller setpoint voltage (necessary when the axis moves in the mechanically incorrect direction). Either bit 1 **or** bit 2 must be changed when the position control direction is inverted. Both bits must be modified if the position control direction is correct but the traversing direction incorrect.

Bit 0 Bit 0=0 Start of reference point approach with ”+” direction key
Bit 0=1 Start with ”-”

568*							
7	6	5	4	3	2	1	0
AXIS NAME							

Standard machine data:

SINUMERIK 810 GA3M/820 GA3M: ”X”, ”Y”, ”Z”
SINUMERIK 810 GA3T/820 GA3T : ”X”, ”Z”, ”Q1”, ”Q2”
(Q1 and Q2 are not available as a standard feature.)

Variable address names

SINUMERIK 810 GA3M: The input keys for axes 1 to 4 are assigned the address letters from machine data 5680 to 5683.

SINUMERIK 810 GA3T: The input keys for axes 1 to 4 are assigned the address letters from machine data 5680 or 5683 as follows:

NC MD 5680 Input key 7
NC MD 5681 Input key 9
NC MD 5682 Input key 8
NC MD 5683 Input key 4th axis.

SINUMERIK 810 GA3 M/810 GA3 T:

For the display of the axis addresses in the actual value displays and programming of axis addresses in part programs, it is necessary to input the axis addresses into machine data MD 5680 ... MD 5687 for the actual and fictitious axes.

MD 5680	Axis 1
:	
MD 5686	Axis 7

The marking of the key caps on the keypad and on the machine control panel remains ”X”, ”Y”, ”Z”, ”4” for SINUMERIK 810 GA3 M and ”X”, ”Z”, ”C”, ”Q” for SINUMERIK 810 GA3 T.

Coding of addresses (axis names)

7 … 4	3 … 0
Extended addresses (address number)	Address name

Bit No.

Extended addresses (address number)	Address name
blank = 0000	X= 0000
1 = 0001	Y= 0001
2 = 0010	Z= 0010
3 = 0011	A= 0011
4 = 0100	B= 0100
5 = 0101	C= 0101
6 = 0110	U= 0110
7 = 0111	V= 0111
8 = 1000	W= 1000
9 = 1001	Q= 1001
	E=1010

e. g. 0000 0010 = Z
0001 1001 = Q1
0010 1001 = Q2

The axis names for all axes existing in one control must be different.

572*							
7	6	5	4	3	2	1	0
			Correction for rotary axis	TO mirroring for transverse axis	Modulo 360 program. for rotary axis	Transverse axis	Zero offset also to be mirrored with ”Mirror axis”

Bit 4=0 All current offsets (e. g. ZO. TO...) are added according to the Modulo calculation (G90, G68) in block decoding for programmed traversing. Axial traversing paths of greater than 360° might arise.

Bit 4=1 ”First traversing movement within a full or semicircle for a rotary axis”

This bit is active only if 572* Bit 2=1 .
When bit 4=1 another modulo calculation is performed after calculation of the offsets (Z0, T0 ...). This gives a traverse movement within 360° in the programmed direction for programming G90. In G68 an additional calculation of +/- 180° is performed in order to traverse the shorter path, i.e. the direction of travel is determined at this point.

Example:

conditions

– MD 560*. 7=1
– MD 564*. 5=1
– MD 572*. 2=1

```
N15  G58  C300  LF
N20  G0   G90   C300  LF
N25  G1   G90   C-200 F1000 LF
N30  G1   C300  LF
N35  G53  G0    C0    LF
N40  M30  LF
```

Path traversed with:		
Block no.	Bit 4=0	Bit 4=1
15	0°	0°
20	600°	240°
25	–100°	–100°
30	100°	100°
35	–240°	–240°
40	–	–

When bit 4=1 only traversing paths within 360° result (N25).

Additional function: Bit 4 must be set if, for rotary axes, clear distance-to-go and read actual value are used with @360.

@360 R... K..
.
.
.
G90 C=R...

Bit 3 If the transverse axis is mirrored, this MD can be used to select whether the TO is also to be mirrored (see also Interface Description Part 1, ”Mirroring”).

Bit 2 Bit 2=1 The guidelines for modulo programming of rotary axes apply (see Section 10.4).

Bit 1 If an axis is defined as a transverse axis, special functions can be activated:

- MD 5011 bit 6 Actual-value display over diameter
- MD 5011 bit 4 Diameter programming for G90
- MD 5011 bit 2 Handwheel/DRF over diameter
- TO wear over diameter
- PRESET over diameter
- MD 572* bit 3

Bit 0 If bit 0 is set, the zero offsets are also mirrored with active PLC signal ”Mirroring” and activated axis duplication.

576*							
7	6	5	4	3	2	1	0
Axis number to which duplication takes place							

Bit 7, 6, 5 Specification of axis number of duplicated axis of tool system 2.
The bits take effect only after ”POWER ON”.

Example:

MD 5680 Axis name 1st axis = X2
MD 5681 Axis name 2nd axis = Z
MD 5682 Axis name 3rd axis = C
MD 5683 Axis name 4th axis = Z2
MD 5684 Axis name 5th axis = X1
MD 5685
MD 5868

MD 5760	Bit 7, 6, 5 = 0
MD 5761	Bit 7, 6, 5 = 100 binary 4th axis (Z2) belongs to tool system 2
MD 5762	Bit 7, 6, 5 = 0
MD 5763	Bit 7, 6, 5 = 0
MD 5764	Bit 7, 6, 5 = 001 binary 1st axis (X2) belongs to tool system 2
MD 5765	
MD 5766	

The following machine data must be identical for both axes when the ”Axis duplication” option is set:

MD	344*	Modulo value for rotary axes
MD bit	560*.2	Rounding full/half degrees
MD bit	560*.3	Rounding for rotary axes
MD bit	560*.5	Software limit switch effective
MD bit	560*.7	Actual value display modulo 360 degrees
MD bit	564*.5	Rotary axis
MD bit	572*.1	Transverse axis
MD bit	572*.2	Program modulo 360 degrees
MD bit	572*.3	Mirror tool offset for transverse axis
MD bit	572*.4	Rotary axis full/semicircle programming

580*

7	6	5	4	3	2	1	0
	Axis with distance-coded reference marks	Position value modulo at Power On		Absolute offset valid	Measuring system polarity oppos. to machine syst. polarity		Axis with SIPOS-absolute encoder

Bit 6 = 1 For this axis the NC machine data MD 384* ”Graduation period of linear scale” and MD 392* ”Absolute offset” must be entered. Reference point approach is performed only by traversing via 2 reference marks. Only the reference point cutoff speed stored in MD 284* is used as reference point speed. If the bit is set, reference point approach is suppressed.

Bit 4 = 1 Position value at POWER ON modulo MD 344*.
This allows the entire encoder range of the SIPOS absolute encoder to be used for rotary axes.

Bit 3 = 1 With bit 3=1 the value stored in MD 392* is declared valid (only for RPC, for SIPOS see decription on next page). On completion of reference point approach, the siganl ”Reference point reached” is output. If bit 3=0, any desired reference point approach is carried out, but the feedback signal ”Reference point reached” is not given.

Bit 2 = 1 The absolute value encoder increases with decreasing machine absolute value and decreases with increasing machine absolute value.

Bit 2 = 0 The absolute value encoder increases with increasing machine absolute value and decreases with decreasing machine absolute value.

Bit 0 = 1 The axis is equipped with SIPOS absolute value encoder.

Like every standard encoder, the SIPOS absolute encoder has the tracks A, B and zero pulse. This allows synchronisation by linking the zero mark with the reference point cam, as in the incremental system.

If MD 580* bit 0=1 ”Axis with SIPOS absolute encoder” is set, there are two possible cases for reference point approach:

Case 1: NC MD 580* bit 3=0

If the ”Absolute offset valid” bit is not set, reference point approach is carried out in the same way as for an axis without absolute encoder. With ”Reference point reached”, the calculated absolute offset is transferred to the axial MD 392*, and MD 580* bit 3 ”Absolute offset valid” is automatically set for the relevant axis.

The absolute offset is calculated according to the equation:

Machine system = SIPOS absolute encoder value+absolute offset, or
Absolute offset = machine system - SIPOS absolute encoder value

Here:

- Machine system = desired absolute position=reference point value (MD 240*)
- SIPOS absolute encoder value = displayed absolute position (actual value)

Note:

If a value other than zero was contained in NC MD 392* this value must be taken into account because it is contained in the SIPOS absolute encoder system (=displayed actual value).

Suggestion:

Set NC MD 580*=0 prior to synchronisation and perform POWER ON.

Case 2: NC MD 580* bit 3=1

If the ”Absolute offset valid” bit is set, reference point approach is suppressed. This applies until the bit is reset by the user.

With the G74 ”Reference point approach from part program” function, reference point approach is not carried out for the 2nd case and processing is continued with the next block.

Note:

Reference point approach is possible again after NC MD 580* bit 3 has been cancelled.

Setup without reference piont

No reference cam is available. MD 580* bit 0=1 is set, i. e. SIPOS absolute encoder is available. The machine is moved manually to the position the customer considers to be the reference point in ”JOG” or ”Incremental” mode. The reference point value is entered inMD 240*.

Calculation of absolute offset:

Machine system = SIPOS absolute encoder value+absolute offset, or

Absolute offset = machine system - SIPOS absolute encoder value

Here:

- Machine system = desired absolute position=reference point value
- SIPOS absolute encoder value = displayed absolute position (actual value)

Note:

If NC MD 392* contained a value other than zero, this value must be taken into account because it is contained in the SIPOS absolute encoder system (=displayed actual value).

When setting up the SIPOS absolute encoder by means of reference point approach, overrun processing is automatically performed.

Overrun occurs because the absolute encoder is absolute to within $\pm 2^{15}$ revolutions and when setting up an overrun can occur when traversing from the zero position to 2^{15} revolutions.

When setting up is performed without reference point, overrun processing must be carried out by the customer. Make sure, in this case, that all absolute values used have the format ±99999999 input units.

If the calculation of absolute offset by the SIPOS absolute encoder value of the machine system yields a value greater or less than±99999999, this value must be corrected as follows:

Absolute offset	>	99999999:	Absolute offset	-	199999999
Absolute offset		-99999999:	Absolute offset	+	199999999

Example:

a) Calculated absolute value=130000000
Correction: absolute offset=130000000-199999999=-69999999
b) Calculated absolute offset=-130000000
Correction: absolute offset=-130000000+199999999=+69999999

Behaviour on warm restart (POWER ON)

If MD 580* bit 0 ”Axis with absolute encoder” is set, MD 580* bit 3 ”Absolute offset valid” is checked for the relevant axis. If both bits are set, the axial interface signal ”Reference point reached” is already set on warm restart.

Special case ”Parking axis”

The axial interface signal ”Parking axis” is used to cancel the interface signal ”Reference point reached” even for an axis with SIPOS absolute encoder. If NC MD 580* bit 3 is set, repeated reference point approach is suppressed. The absolute value is not transferred until POWER ON is actuated.

Absolute encoder error

All errors of the absolute encoder are displayed in the control by means of an error message (Error no. 41 - absolute submodule error).

The type of error can be seen from the line ”Absolute submodule status” in the service data axis menu. The error number is displayed.

Note:

- When the absolute encoder is switched on for the first time, different absolute encoder error numbers are displayed for a period of approx. 10 minutes. This is due to the backup battery in the absolute encoder which needs a certain time to be charged up until error-free communication between control and absolute encoder is possible.
- All absolute encoder errors are signalled by the absolute encoder while the control ramps up. Acknowledgement of the absolute encoder errors (e. g. by the Reset key) is not possible. Elimination of errors, i. e. faultless operation, cannot be signalled to the control until repeated POWER ON.

Remarks

- MD 580* bit 3 =1
  - Software limit switches and leadscrew error compensation are enabled (if available).
  - The leadscrew errors between reference point and absolute position are taken into account.
- Reference point approach with SIPOS absolute encoder is not possible until MD 580* bit 3 has been reset.
- Whenever the absolute value is calculated without reference point approach, the control must be informed on whether or not the SIPOS absolute encoder has the same polarity as the machine system. Whether or not the SIPOS absolute encoder has the same polarity can be easily determined after synchronisation. The relevant axis is moved to an arbitrary position and the control is started.The system has the same polarity and need not be corrected if after setting up, positive traversing and POWER ON a greater absolute encoder value is displayed. If a smaller absolute encoder value is displayed, MD 580* bit 2=1 must be set. The display shows the correct actual position after the next POWER ON.
- MD 580* bit 2 must also be adapted when the mechanical travel direction is changed with MD 564*.

584*	Resolutions						
7	6	5	4	3	2	1	0
Display resolution					Position control resolution		

The display resolution determines the incremental weighting for the dimensional displays.
The reset position G70 (inch) or G71 (mm) is defined in MD 5002 bit 4 for both input resolution and display resolution. This applies for all axes.
The position control resolution (ms) determines the allocation of an increment of the partial actual value to a specific traversing path.
Whether the measuring system is metric or inch is defined in MD 5002 bit 0 for all axes together.

Standard value: 01000010

Coding of the resolutions and the possible combinations of input, position control and display resolutions are described in Section 10.7 ”Difference input, position control and display resolutions”.

6000*	Compensation points for leadscrew error compensation						
7	6	5	4	3	2	1	0
Comp. point 4		Comp. point 3		Comp. point 2		Comp. point 1	
yes/no	+/-	yes/no	+/-	yes/no	+/-	yes/no	+/-

6248*	Compensation points for leadscrew error compensation						
7	6	5	4	3	2	1	0
Comp. point 996		Comp. point 995		Comp. point 994		Comp. point 993	
yes/no	+/-	yes/no	+/-	yes/no	+/-	yes/no	+/-

6249*	Compensation points for leadscrew error compensation						
7	6	5	4	3	2	1	0
Comp. point 1000		Comp. point 999		Comp. point 994		Comp. point 993	
yes/no	+/-	yes/no	+/-	yes/no	+/-	yes/no	+/-

– = 0 =negative compensation
+ = 1 =positive compensation
no = 0 =no compensation
yes = 1 =compensation active

Note:

MD Nos. 6000 to 6249 are only used for the leadscrew error compensation option. See Section 10.3 for a description of this function.

1096*	Fictitious 2nd reference point		
Standard value	Lower input limit	Upper input limit	Units
0	- 99 999 999	99 999 999	units (MS)

*= 0= 1st axis
*= 1= 2nd axis
*= 2= 3rd axis
*= 3= 4th axis
*= 4= 5th axis
*= 5= 6th axis
*= 6= 7th axis

A ”second reference point” can be activated by calculating the difference of the two reference point values (MD 240*, 1096*) and by entering these in the external zero offset via the PLC program. This is a very practical approach for the complete machining of workpieces on turning machines, for example.

9 Setting Data

9.1 Softkey assignment

Softkey:	”ZERO OFFSET”	Section 9.2
Softkey:	”PROGR. EXT. ZO”	Section 9.3
Softkey:	”ZO ADDIT.”	Section 9.4
Softkey:	”R PARAMETERS”	Section 9.5
Softkey:	”SPINDLE”	Section 9.6
Softkey:	”AXIAL”	Section 9.7
Softkey:	”SETTING BITS”	Section 9.8
Softkey:	”ROTAT. ANGLE”	Section 9.10
Softkey:	”SCALE MODIF.”	Section 9.11

All setting data (SD) are active immediately (without Power On).

The general and RS232C (V.24) SD bits (Section 8.2.6) may also be modified in ”SET UP OVERALL RESET” mode.

9.2 Zero offset

G54	ZO 1
G55	ZO 2
G56	ZO 3
G57	ZO 4

The basic setting for programming is G54.

Block-by-block cancellation of all ZO using G53.

Note:

For axis duplication G54 to G57 are axially active.

9.3 Programmable ZO + external ZO

G58	1st programmable ZO
G59	2nd programmable ZO
Ext. ZO	External additive ZO The values are transmitted from the PLC via external data input. They may only be deleted by the PLC or using the ”FORMAT USER MEM.” softkey.

The ”External data input” option must be available.

Block-by-block cancellation of all ZO using G53.

Note:

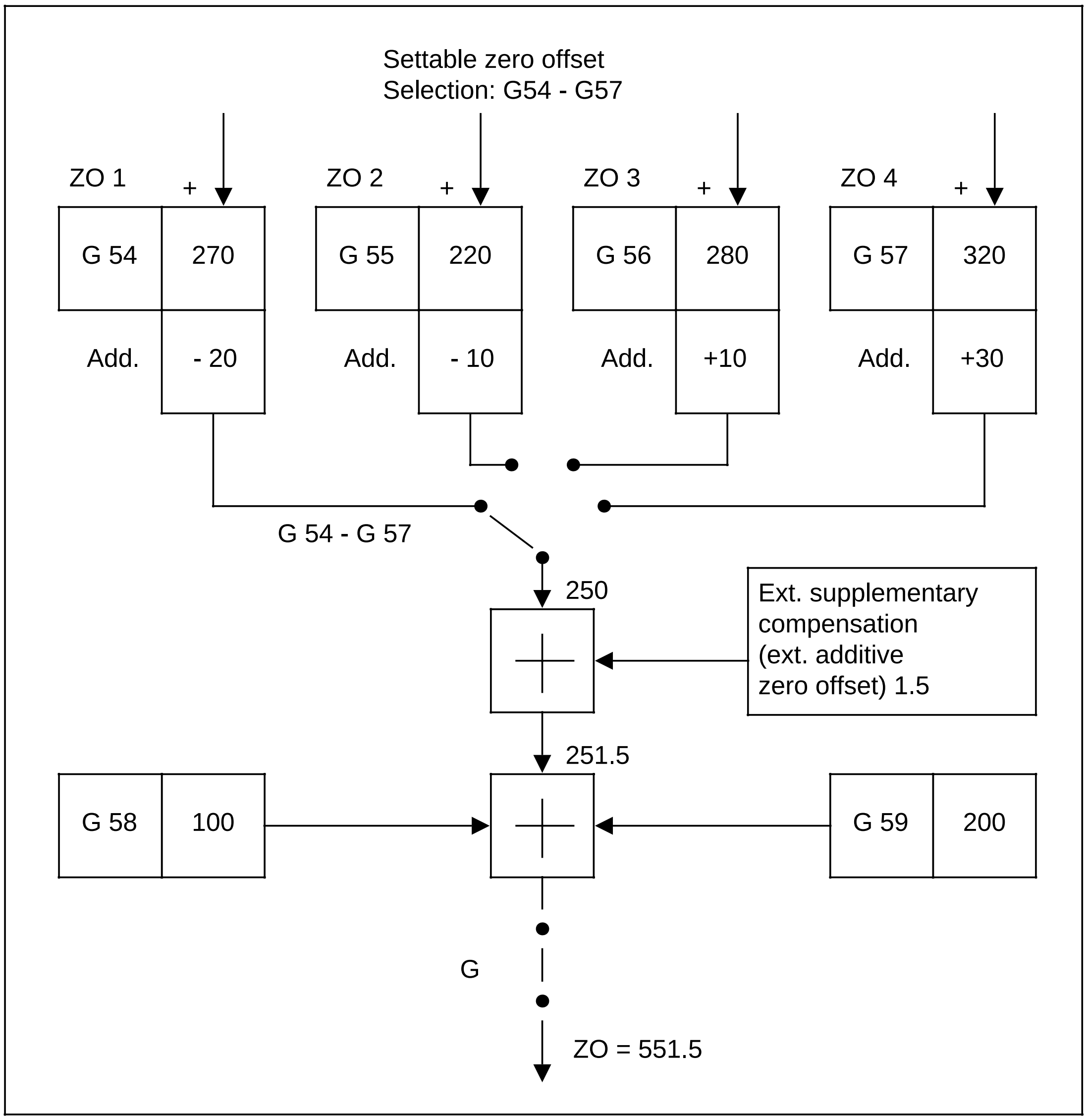
For axis duplication exteral zero offsets remains axially effective whereas G58 and G59 are ”duplicated” to the associated axes in tool system 2. In the ”Programmed zero offsets” display, the zero offset is always displayed for both axes (programmed and duplicated axis) irrespective of the active PLC signals Q 81.2, Q 81.3.

9.4 Additive ZO

G54 1st additive ZO
G55 2nd additive ZO
G56 3rd additive ZO
G57 4th additive ZO

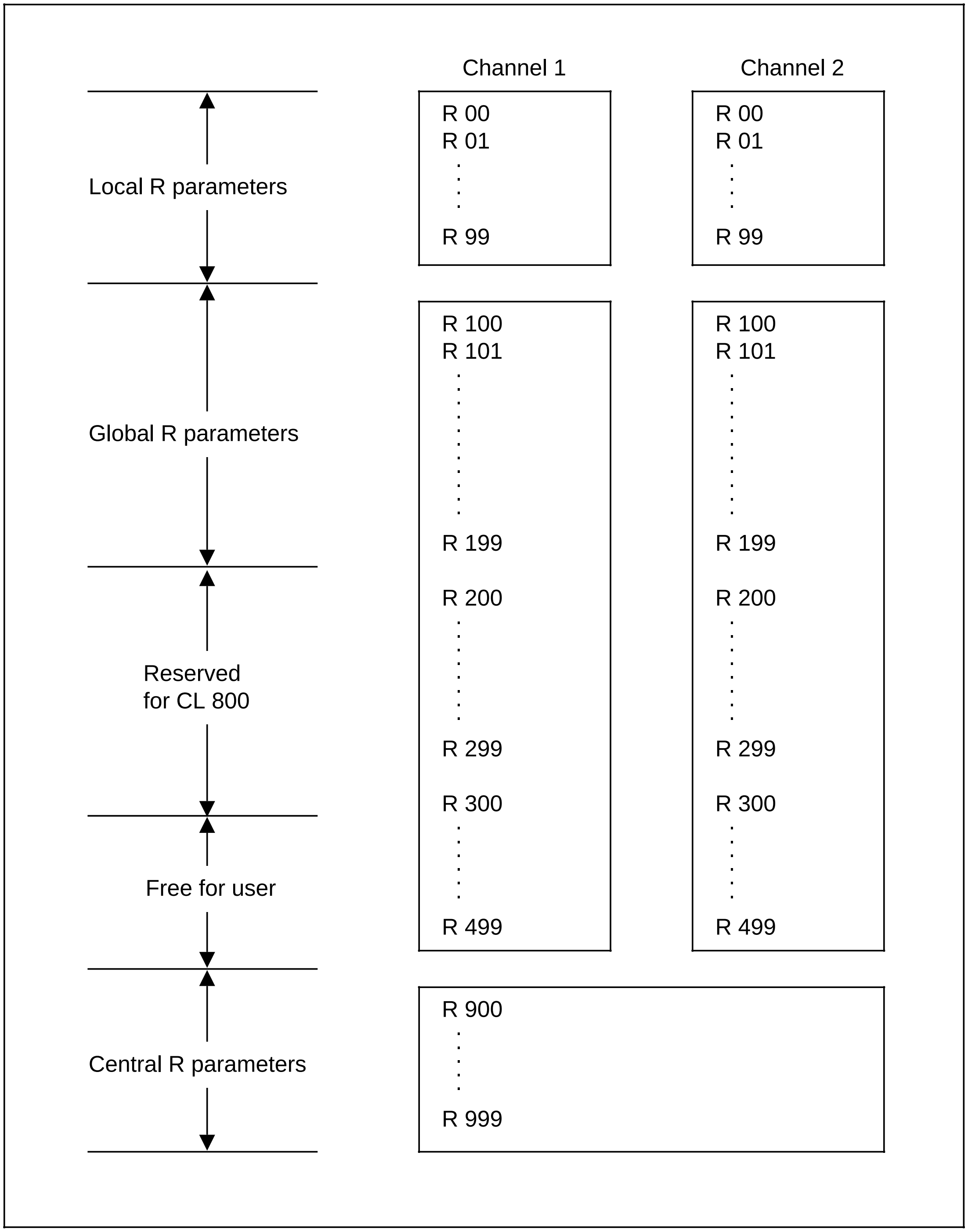
Not only the ZO (Section 9.2) but also the associated additive ZO are activated by means of G54-G57 in the program.

The basic setting for programming is G54.



9.5 R parameters

Parameters R000 - R699 are channel-specific, i.e. additionally available for each channel.
Parameters R700 - R999 are available for all channels together.



Breakdown of R parameters

R 0 ... R 49:	Transfer parameters for SIEMENS cycles. Available to the user if these cycles are not used.
R 50 ... R 99:	Local R parameters in the SIEMENS cycles. Saved by these cycles via PUSH/POP. Available to the user.
R 100 ... R 109:	Available to the user.
R 110 ... R 199:	Assigned for measuring cycles.
R 200 ... R 249:	Assigned internally.
R 250 ... R 280:	Allocated to cycles with the T version.
R 300 ... R 499:	Assigned internally. R 300 stack pointer for @040 ... 043 R 301 to 499 stack area R 300 must not be employed by the user R 301 to 499 are used by @040 ... 043 and by the cycles.
R 500 ... R 699:	Channel-specific R parameters, available to the user.
R 700 ... R 959:	Central R parameters, available to the user.
R 960 ... R 999:	Assigned to tool management. These central parameters are available to the user if tool management is not used.

9.6 Spindle setting data

Programmed spindle speed limitation

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
401*	**) programmable spindle speed limitation	0	16000		rev/min

Additionally to fixed spindle speed limitation, programmed spindle speed limitation makes it possible to reduce the spindle speed in the program with G function G92 with V constant (G96) selected.

Example:

% 7
N10 ... LF
N20 G92 S3000 LF
N30 ... LF

The possible spindle speed for G96 is limited to S3000. The value is entered automatically in the SD.

* *0=spindle 1, 1=spindle 2*

Oriented spindle stop (M19)

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
402*	Oriented spindle stop	0	3599		1/10 deg.

Positioning to this angle (in degrees) is performed with M19 programmed.

Example:

- With M19 S270 LF the spindle is positioned to 270 degrees and the angle is entered in the SD.
- With M19 LF the spindle is positioned at the angle entered in the SD.

Spindle speed limitation

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
403*	Spindle speed limitation	0	16000		rev/min

With spindle speed limitation, the spindle is limited to the entered speed (0 signifies no spindle rotation possible).

Feed acceleration ramptime for thread

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
404*	Smoothing constant thread	0	5		

This time is used to influence the magnitude of the change in feedrate in the event of a change in spindle speed.

In other words, fluctuations in the spindle speed are averaged over the stated cycle time before they are used by the NC software to influence the feedrate.

Input value	0	1	2	3	4	5
Cycle time x	0	1	3	7	15	31
Setpoint output for feed drive	Jump		Ramp			

With the function ”Tapping with dynamic following error compensation” this data must have the value zero.

9.7 General and axis-specific SD

Dry run feedrate

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
0	Dry run feed	0	44000	IS	1000 units/min

If dry run feedrate is selected at the control, the dry run feedrate (mm/min (G94)) is selected as the tool path speed as opposed to the programmed rate.

The dry run feedrate must be entered before NC START.

Minimum/maximum working area limitation

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
300*	Min. working area limitation [1)]	0	+/–99999.999	IS	mm or inch
304*	Max. working area limitation[1)]	0	+/–99999.999	IS	mm or inch

This setting data makes it possible to limit the traversing ranges in automatic (JOG) mode (in addition to the software limit switches). The working area limits can be modified in the program by means of G25/G26.

Axis/handwheel assignment

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
308*	Axis/handwheel assignment	0	0...3	– –	– –

This is used to specify the handwheel for operation of the axes
(1=handwheel 1, 2=handwheel 2, 3=handwheel 3)

9.8 General and RS232C (V.24) - SD bits

SD 5000 Bit 0,1,2: As from standard UMS 03 the user interface and the cycles were completely revised. The cycles have new and expanded functions. For these functions new or modified cycle parameters (R parameters) are required.

To guarantee part program compatibility new functions with set SD bits are activated.

For programs developed with older software versions (e.g. 810 A1 SW06), the SD bits must be zero (compatible mode).

For programs developed with user interface above UMS03, the SD bits must be set accordingly.

		Turning cycles L95/L93/L98	Drill patterns Milling patterns L903/L930	Drill cycles L81 - L89
SD 5000	Bit	2	1	0
	M	0	1	1
	T	1	0	1

Bit = 1 cycle function expansion by UMS from version 03 can be used
Bit = 0 cycle functions such as UMS version 02 or below (compatible mode)

SD 5001 Bit 0=1: Display of workpiece-related actual-position system
The actual-position display refers to the workpiece zero and not to the reference point, i.e. zero offsets and tool offsets are not displayed.

SD 5002 Bit 5=1: Part program overview with comment's block. The first comment in the part program is displayed in the part program overview.

SD 5010 ... 5029: 1st + 2nd RS232C (V.24)
The description is included in the configuring instructions (Universal Interface).

SD 560* Axis-specific bits

Bits for 1st serial interface

SD No.	Bit No. 7	6	5	4	3	2	1	0
5010	Device ID 1st interface, read in							
5011	Transfer format 1st interface, read in							
	No. of stop bits		odd parity	with parity			Baud rate	
5012	Device ID 1st interface, read out							
5013	Transfer format 1st interface, read out							
	No. of stop bits		odd parity	with parity			Baud rate	
5014	Xon start character 1st interface (value e.g. . 11H)							
5015	Xoff start character 1st interface (value e.g. . 93H)							
5016	Special bits 1st interface							
	Start without Xon	Prog. start with LF	End of block with CR LF	Output in EIA code	Stop with end of trans. character	Evaluate ready	No leader and trailer on read out	Prog. system 3/8 read in
5017	Special bits 1st interface							
						Do not output NC MD with 0	No REORG via interface	Time monitoring off

Bits for 2nd serial interface

SD No.	Bit No. 7	6	5	4	3	2	1	0
5018	Device ID 2nd interface, read in							
5019	Transfer format 2nd interface, read in							
	No. of stop bits		odd parity	with parity			Baud rate	
5020	Device ID 2nd interface, read out							
5021	Transfer format 2nd interface, read out							
	No. of stop bits		odd parity	with parity			Baud rate	
5022	Xon start character 2nd interface (value e.g. 11H)							
5023	Xoff start character 2nd interface (value e.g. . 93H)							
5024	Special bits 2nd interface							
	Start without Xon	Prog. start with LF	End of block with CR LF	Output in EIA code	Stop with end of trans. character	Evaluate ready	No leader and trailer on read out	Prog. system 3/8 read in
5025	Special bits 2nd interface							
						Do not output NC MD with 0	No REORG via interface	Time monitoring off

Common bits

SD No.	Bit No. 7	6	5	4	3	2	1	0
5026	EIA code for "@" (value e.g. 6DH)							
5027	EIA code for ":" (value e.g. 46H)							
5028	End of transmission character (value e.g. 03H)							
5029	EIA code for "="							

Baud rate table

Bit No. 3	2	1	0	Baud rate
0	0	0	0	110 baud
0	0	0	1	150 baud
0	0	1	0	300 baud
0	0	1	1	600 baud
0	1	0	0	1200 baud
0	1	0	1	2400 baud
0	1	1	0	4800 baud
0	1	1	1	9600 baud

Setting data for PLC programmer (PG615, PG670, PG675, PG685, PG750)

5010	00000100	**5011**	xxxxx111

Note:
For reading in/out alarm texts, the setting data 5010 for interface 1 and 5018 for interface 2 must be adapted as follows for the languages "Hungarian, Czech and Turkish":

00001001 must be specified for the variant 8BIT-XONOFF
00001000 must be specified for the variant 8BIT-RTS.

Setting data interfaces							
1st interface	**5010**	**5011**	**5012**	**5013**	**5014**	**5015**	**5016**
2nd interface	**5018**	**5019**	**5020**	**5021**	**5022**	**5023**	**5024**
PG 675/685/750/ (CP/M86 1200 baud) RS232C (V.24) 9600 baud	00000000	11000100 or 11000111	00000000	11000100 or 11000111	xxxxxxxx	xxxxxxxx	xx1x1xxx
GNT reader (optional B02/B03)	00000000	11000111	xxxxxxxx	xxxxxxxx	xxxxxxxx	xxxxxxxx	00000000
Siemens reader (Fanuc)	00000010	xxxxxxxx	xxxxxxxx	xxxxxxxx	xxxxxxxx	xxxxxxxx	xxxxxxxx
Fanuc hand-held reader	00000001	11000110	xxxxxxxx	xxxxxxxx	00010001	10010011	00000000
PT80 300 baud	00000000	11000010	00000000	11000010	xxxxxxxx	xxxxxxxx	00000000
PT88 9600 baud RS232C (V.24)	xxxxxxxx	xxxxxxxx	00000000	11000111	xxxxxxxx	xxxxxxxx	00000000

Excerpt from the universal interface. ”x” = bit setting optional.
SD 5018 and SD 5028 have no significance here.

For ISO, EVEN parity must be coded in the freely definable special characters Xon and Xoff, and for EIA, ODD parity.

9.9 Axis-specific bits

SD No.	Bit No. 7	6	5	4	3	2	1	0
560*						Enable scale modification		
562*								

With SD 560*, bit 2 an axis is enabled for scale modification.

9.10 Angle of rotation, for channels 1 and 3 separately

G54 A= Settable
G55 A=
G56 A=
G57 A=

G58 A= Programmable
G59 A=

For channel 2 no coordinate rotation is possible.

9.11 Scale modification, for channels 1 and 3 separately

Only programmable.

Scale factor

Channel 1 P (P=1 means no scale modification)
Channel 3 P

Scale centre separate in each axis.

Programming: G51 X Y Z P

P — Scale factor

X Y Z — Scale centre

Enabling of the axis for the function scale modification in SD 560*, bit 2.

Note:

Setting data bits of tool system 1 are effective with activated ordering data extension Axis Duplication.

10 Description of Operation

10.1 Spindle control

10.1.1 Corresponding MD

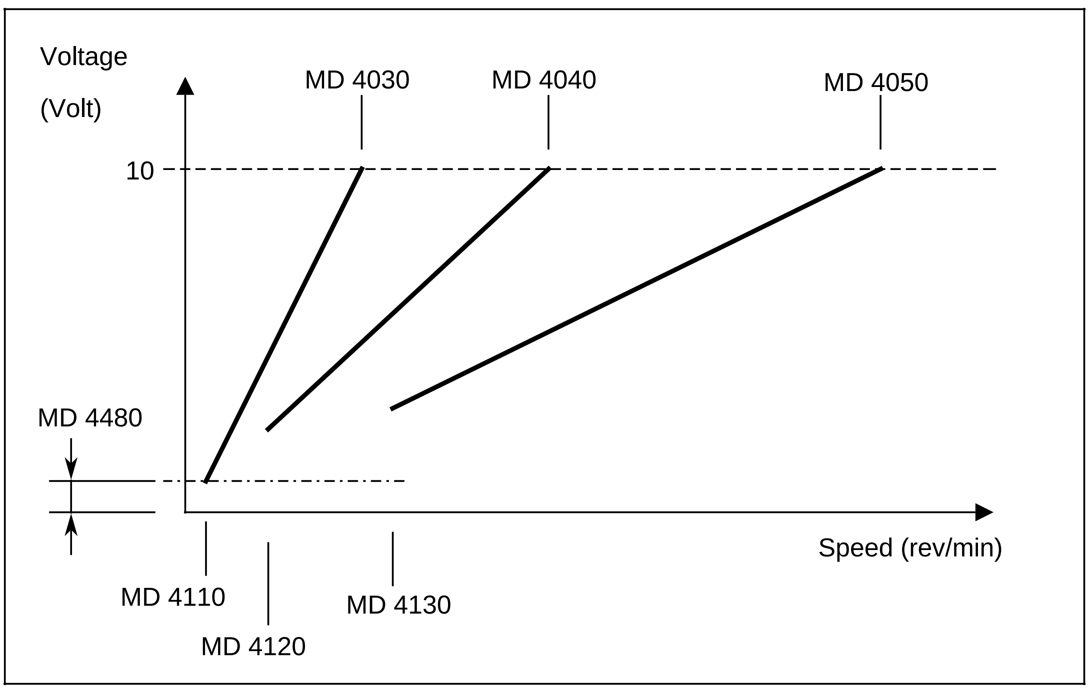
- MD 131 ... 146 (Spindle override)
- MD 400* ... 452* (Spindle data)
- Option ANALOG SPINDLE SPEED
- Option 2nd spindle
- MD 520* Bit 0 to 7
- MD 521* Bit 1 to 7

10.1.2 S analog (Mn=3, Mn=4, Mn=5)

Spindle commissioning is explained in Section 5.3.

In the case of the SINUMERIK 810 GA3/ 820 GA3, output of the analog spindle speed is fully implemented in the NC, so the PLC can only be influenced by means of special signals (see Section 10.1.4).

The "Automatic gear selection" function is also implemented in the NC, the correct gear being transmitted to the PLC as a function of the programmed S value. The speed range of the max. 8 gears is defined by entering the minimum and maximum speed in the relevant NC MDs.



Determining the speed range using min. and max. speed (Example for spindle 1)

---

n: 0 = leading spindle
1 = spindle 1
2 = spindle 2

The gear is output on the basis of the lowest switching frequency, i.e. if the speeds of the individual gears overlap, a new gear is only output when the programmed S value is no longer possible in the selected gear.

In view of the fact that not all spindle drive units include a ramp-function generator, such a generator was integrated into the SINUMERIK 810 GA3/820 GA3 (4 ms units). The following enable signals are required for spindle setpoint output:

• Q 100.7**/**Q 104.7	SPINDLE ENABLE
• Q 100.6**/**Q 104.6	SPINDLE SERVO ENABLE
• Setting data	SPINDLE SPEED LIMITATION
• Q 100.5=0**/**Q 104.5 = 0	SET SETPOINT ZERO
• Q 100.4=”1”**/**Q 104.4 = 1	SPINDLE SPEED OVERRIDE ACTIVE

Setpoint output is controlled using M functions M3, M4, M5. In the case of M5 no setpoint is output (drift remains) but the spindle servo enable relay does not drop out, i.e. the spindle may drift if Q 100.6 is **not** cancelled.

(n=1 for spindle 1; if there is only one spindle, n = need not be written)
(n=2 for spindle 2)

10.1.3 Mn=19 (oriented spindle stop)

The oriented spindle stop function (Mn=19 Sn=.. LF) is intended to prevent additional external hard-ware requirements if the spindle is to be stopped in a specific position for a tool change or to engage gear.

In this case the ROD encoder is used not only for speed control purposes (G95, G96, G97) and for thread cutting (G33) but also as a position sensor, the zero mark serving as the position reference point (corresponding to 0°).

With S analog (Mn=3, Mn=4, Mn=5) the spindle is controlled by the NC; only with Mn=19 is the position control loop closed by the NC. The pulses from the ROD encoder act as actual position val-ues. When a ROD encoder with 1024 or 512 pulses per revolution is used as the actual-value system, a resolution of 360°**/**4096 (approx. 1**/**11 degrees) can be achieved through four fold hardware multiplication (only 2**/**11 degrees in the case of a ROD encoder with 512 pulses).

The oriented spindle stop function is activated in the part program with ”Mn=19”. The target position is stored as a setting data which can be set with manual input or ”Mn=19 Sn=...” pro-gramming in half degrees.

The positioning range is 0.1 to 359.9 degrees.

Positioning is carried out in the specified direction of rotation (Mn=3, Mn=4) or from rest over the shortest travel.

The spindle may also be positioned using external devices if the ”ORIENTED SPINDLE STOP” option has not been set. In this case, Mn=19 is output to the PLC as a normal auxiliary function (also a static or dynamic flag).

MD 520* bit 2 ”pulse generator available” has no significance.

---
n: 0 = leading spindle
1 = spindle 1
2 = spindle 2

In the case of the **NC-internal** approach, there are two sequences (sequence A or sequence B) on the basis of which oriented spindle stop is integrated into the block sequence of the NC program.

With **sequence A** spindle stop is handled in a special part program block and a block change is not performed until the operation has terminated; axis movements at the same time as spindle positioning are not possible.

With **sequence B** M19 is modal, even over a number of blocks. While the spindle is being positioned or held in position closed-loop control, the axes can be moved, the program further processed or a tool change may even be performed.

The following applies to both sequences:

- Spindle servo enable (Q 100.6 or Q 104.6) must be present prior to Mn=19.
- Mn=19 S=... must be programmed in a special block without axis movements.
- Orientation is performed in the specified direction of rotation (Mn=03/Mn=04).
- Mn=19 is possible from rest (shortest travel).
- Oriented spindle stop is initiated at the start of the block.
- Mn=19 is aborted or terminated by means of
  - Cancel spindle enable (Q 100.7 or Q104.7), if NC MD 521* bit 4 = 0
  - Mn=03 or Mn=04 in NC part program
  - Interface signal ”ACKNOWLEDGE M19” (Q 103.2 or Q 107.2) when SPINDLE CONTROL is on (Q 103.0 or Q 107.0) and NC MD 5210 bit 4 = 1
  - RESET, program end (M02/ M30) (depending on MD 520*, bit 6 (no M19 abort at RESET)
  - EMERGENCY STOP
  - Measuring-circuit error
  - Faults resulting in shutdown of all axes (NC Ready 2 cancelled)
- in OVERSTORE it is possible to input Mn=19, S=... which can then be started with NC START.
- M19 is output as an auxiliary function at the PLC interface (static or dynamic flags)

Special features of sequence A (NC MD 520* bit 5=0) Mn = 19 without axis movem.

- A block change is performed on completion of the Mn=19 function.
- Simultaneous traversing of the axes is not possible.
- A mode change during Mn=19 is not possible.

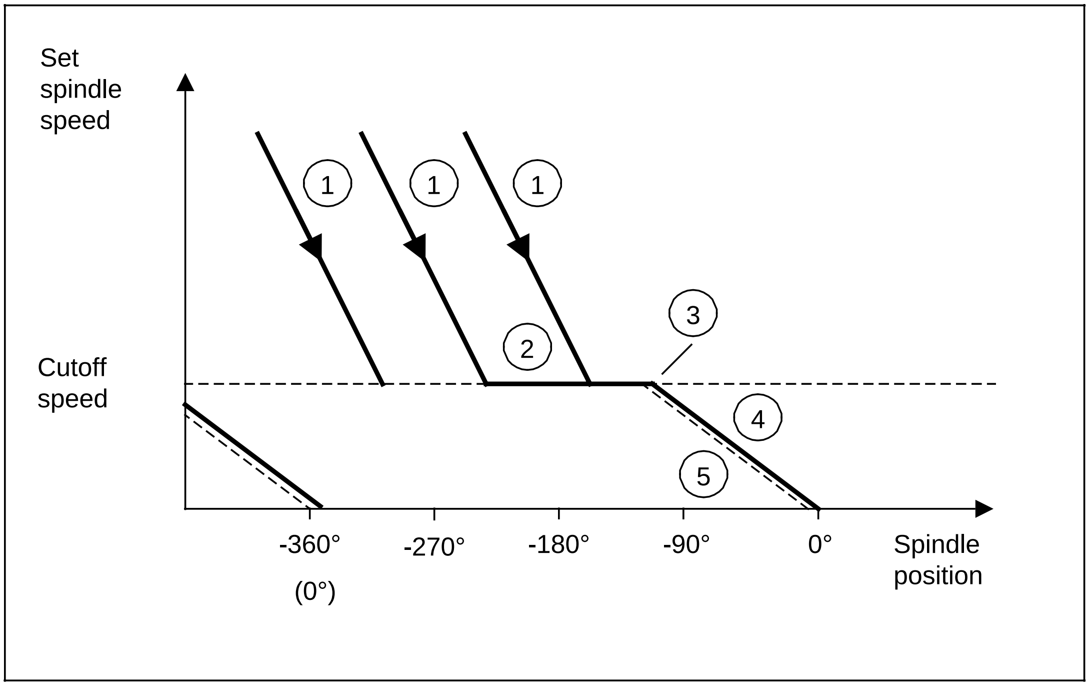
Special features of sequence B (NC MD 520* bit 5=1) Mn=19 with axis movement

- M19 is modal, even over a number of blocks
- M19 is terminated asynchronously to the part program by
  - Cancellation of spindle enable (Q100.7 or Q 104.7), if NC MD 5210 bit 4 = 0
  - Issuing signal ACKNOWLEDGE M19 (Q 103.2 or Q 107.2) PLC spindle control (Q 103.0) must be active and NC MD 521* bit 4 = 1.
- Mn=19 is terminated synchronously to the part program by programming Mn=03 or Mn=04.
- Block change is performed after the delay of one PLC cycle.
- In subsequent blocks axes can be moved or a tool change performed at the same time as positioning or closed-loop position control.
- With active positioning (Mn=19) the direction of rotation (Mn=03/Mn=04) must not be reversed, otherwise positioning is performed from an undetermined direction.
- During Mn=19 the operating mode can be switched;
  the position control loop remains closed and the axes can be traversed.

---

n: 0 = leading spindle
1 = spindle 1
2 = spindle 2

- If the PLC detects auxiliary function M19, it can prevent a block change by cancelling READ-IN ENABLE.
- If Mn=19 is re-selected before a previous Mn=19 has been terminated by cancelling the spindle enable signal, control is performed with respect to the new spindle position, and the NC approaches the new position over the shortest distance, irrespective of the specified direc-tion of rotation; the spindle travel is less than 180°, irrespective of the control characteris-tic.



Control characteristic on repeated selection of Mn=19

a) Decelerating the spindle from the programmed speed to the cutoff speed over the ramp characteristic specified by means of the accelerating time constant .

b) Continue in the same direction of rotation at cutoff speed .

c) Point of intersection of gain characteristic with cutoff speed .
At this point the spindle position controller is switched on together with approach to the programmed spindle position at the gain characteristic .

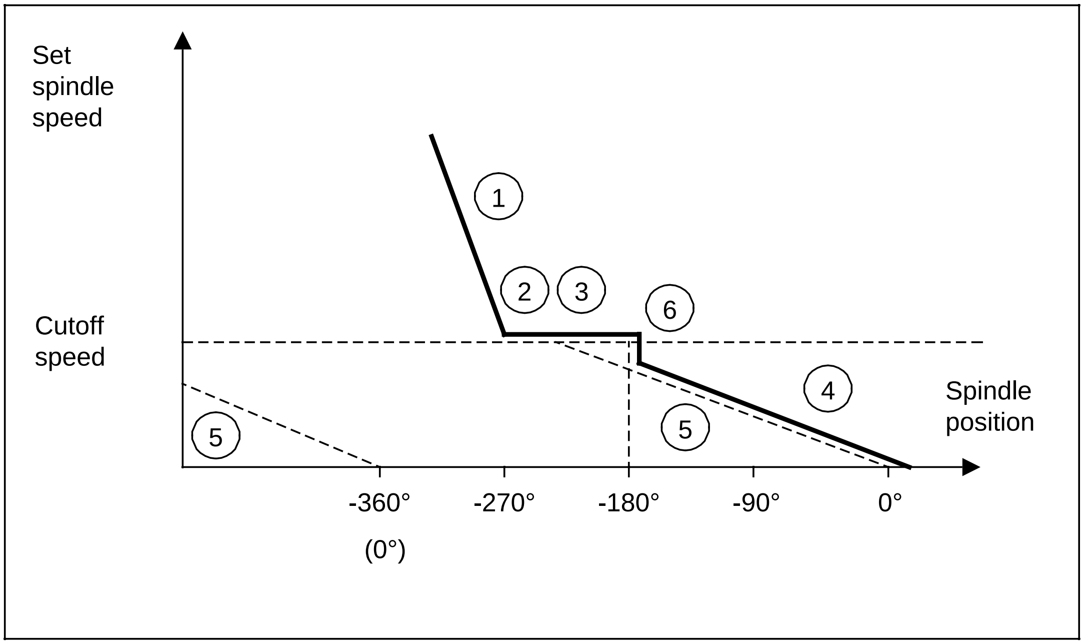
d) Output of message ”SPINDLE POSITION REACHED” (I 114.4 or I 116.4) to the PLC if the actual spindle position has fallen short of the tolerance limit in MD 4430. However, the control continues to try to approach the programmed position with even greater accuracy, as is possible with drift and the slope of the gain characteristic.

e) M19 is terminated when the output signal SPINDLE ENABLE (Q 100.7 or Q 104.7) from the PLC is cancelled while NC MD 521* bit 4 = 0, or when the signal ACKNOWLEDGE M1 (Q 103.2 or Q 107.2) is given depending on NC MD 521* bit 4 = 1 (PLC SPINDLE CONTROL (Q 103.0 or Q 107.0) must be on). In this case the position control loop is opened, but the spindle servo relay does not drop out (the spindle can drift out of position).

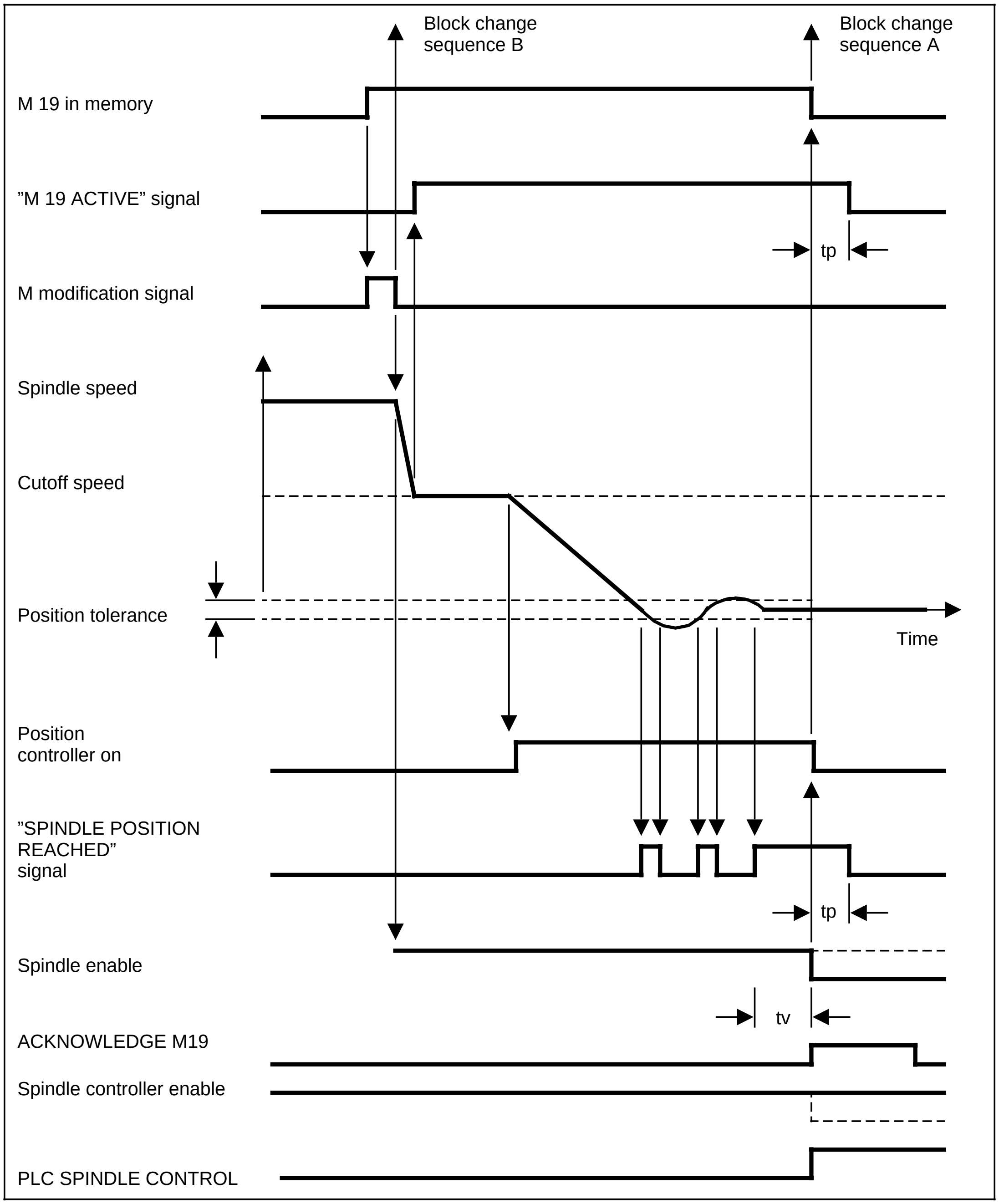
---

n: 0 = leading spindle
1 = spindle 1
2 = spindle 2

If the gain selected is so small that the point of intersection of the gain characteristic with the cutoff speed is more than 180° from the set position, approach to 180° ahead of the set position after the cutoff speed has been reached is performed at the same speed, followed by a jump to the approach curve .



Control charcteristic when small gain is selected



Signal sequence for M19 and NC MD521.4 = 1 (acknowledge M19)*

t_v .... Delay so that positioning is not aborted on overshoot . The delay must be implemented in the PLC program.

t_p .... Time delay, 1 PLC cycle

Mn=19 from rest

In the event of PLC SPINDLE STOP at the start of a block with Mn=19, the function is not immediately considered to be terminated, a block change only being initiated after the delay of one PLC cycle has elapsed.

The PLC can then recognize auxiliary function M19, set spindle enable and initiate spindle orientation from rest. It must be borne in mind that the spindle must be enabled by the PLC while the modification signal (strobe signal) is available. If spindle enable is cancelled late, M19 is taken to be a normal auxiliary function. The spindle will always approach the programmed position from rest over the shortest distance. The ”SET DIRECTION OF ROTATION CLOCKWISE” signal (Q 100.7 or Q 104.7) has no significance.

Positioning accuracy

The target position is programmed in 0.1 degrees with decimal point.
The accuracy of this position depends on the gain and drift. The maximum achievable accuracy is:

Pulse generator, 1024 pulses: $\dfrac{360}{1024 \cdot 4}$ =approx. 0.1 degrees

Pulse generator, 512 pulses: $\dfrac{360}{512 \cdot 4}$ =approx. 0.2 degrees

M19 and RESET

NC-MD 520* bit 6 (no M19 abort on RESET) can be used to prevent function M19 from being aborted with program end (M30/M2) or RESET (key). In this case M19 is aborted only by removing the spindle enable signal or by the ”Acknowledge M19” signal (Q103.2 or Q107.2) from the PLC, by alarms cancelling NC Ready 2, or by EMERGENCY STOP.

Spindle enable with M19

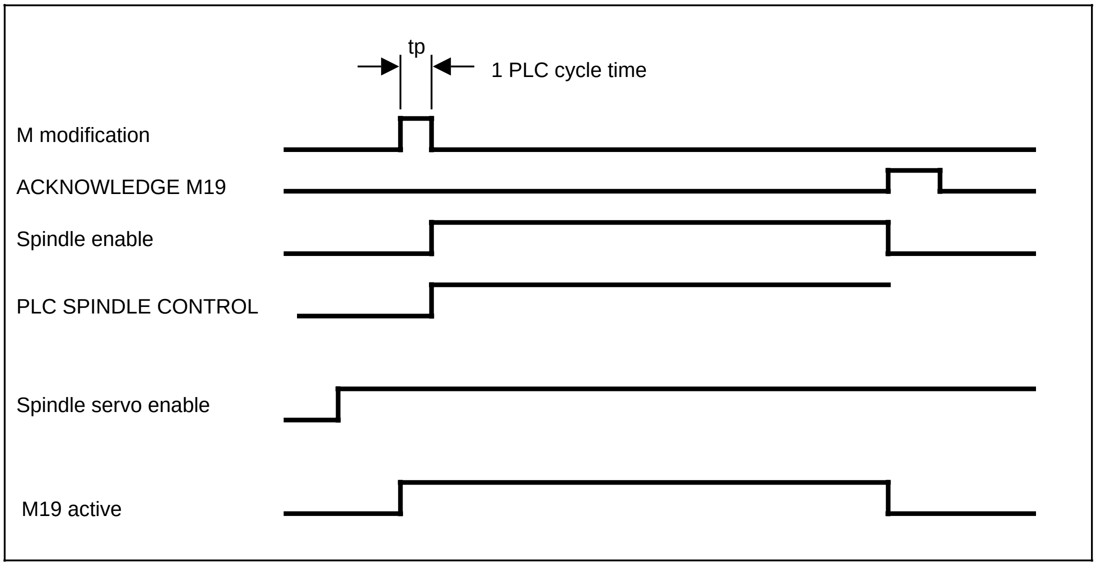
The interface signal SPINDLE ENABLE (Q100.7 or Q 104.7) from the PLC has a second important meaning with M19:

It not only switches the spindle speed set point to zero but also terminates the M19 function (if NC MD 521* bit 4 = 0). M19 is interrupted if spindle enable is not available or has been cancelled while M19 is active.

If the ”SPINDLE POSITION REACHED” signal is output by the NC during positioning, the PLC must cancel the spindle enable signal (if NC MD 521* bit 4 = 0) or issue ”ACKNOWLEDGE M19” (if NC MD 521* bit 4 = 1) if the M19 function is to be terminated. However, spindle enable may only be cancelled if the spindle is in a transient condition. In the event of serious overshoot (very high gain) it is advisable to output the ”SPINDLE ENABLE ” signal to the NC after a delay only when the ”SPINDLE POSITION REACHED” signal is definitely present for a machine-specific time t_V.

Spindle servo enable with M19

Spindle servo enable (Q 100.6 or Q 104.6) cannot be used to control M19. In addition, it must be present prior to M19, i.e. the M modification signal can only be used to set spindle enable, not spindle servo enable.



Control curves of M19 and spindle servo enable

Position control direction with M19

If the spindle is to be switched with M19 from open-loop to closed-loop control, the pulses from the ROD encoder must have the correct directional rotation on reaching the control.

An incorrect position control direction is characterized by the fact that 180° from the programmed position, the spindle exhibits severe reciprocation about this position. In this case NC MD 520* bit 1 must be inverted (actual value sign inverted).

M19 immediately after Power On

As the spindle value is not yet synchronised, the spindle is accelerated to cutout speed. The zero mark of the spindle encoder is acquired and the M19 operation initiated.

Positioning accuracy

The accuracy of the destination presetting is 0.1 degrees. The accuracy depends on the gain, on the gear speed and on the drift. The maximum attainable accuracy is:

For encoders with 1024 pulses: $\frac{360}{1024 \cdot 4}$ = approx. 0.1 degrees

For encoders with 512 pulses: $\frac{360}{512 \cdot 4}$ = approx. 0.2 degrees

An example of the dependency of positioning accuracy on gear speed and gain:

a) Gear speed 1
maximum speed 3000 rev/min = setpoint = 10V
gain e.g. 200 rev/min/360°

$$X = \frac{200}{3000} \cdot 10V = 0.666\ V/360°$$

X = Computed setpoint which the NC would output, if the spindle were 360° out of position.

Smallest voltage unit $1\ DAC = \frac{10V}{8192} = 1.2\ mV$

$$p = \frac{1.2}{666\ mV} \cdot 360° = 0.64°$$ positioning accuracy

b) Gear speed 2
maximum speed 10 000 rev/min = setpoint = 10V
gain e.g. 200 rev/min/360°

$$X = \frac{200}{10000} \cdot 10V = 0.2\ V/360°$$

Smallest voltage unit 1 DAC =1.2 mV

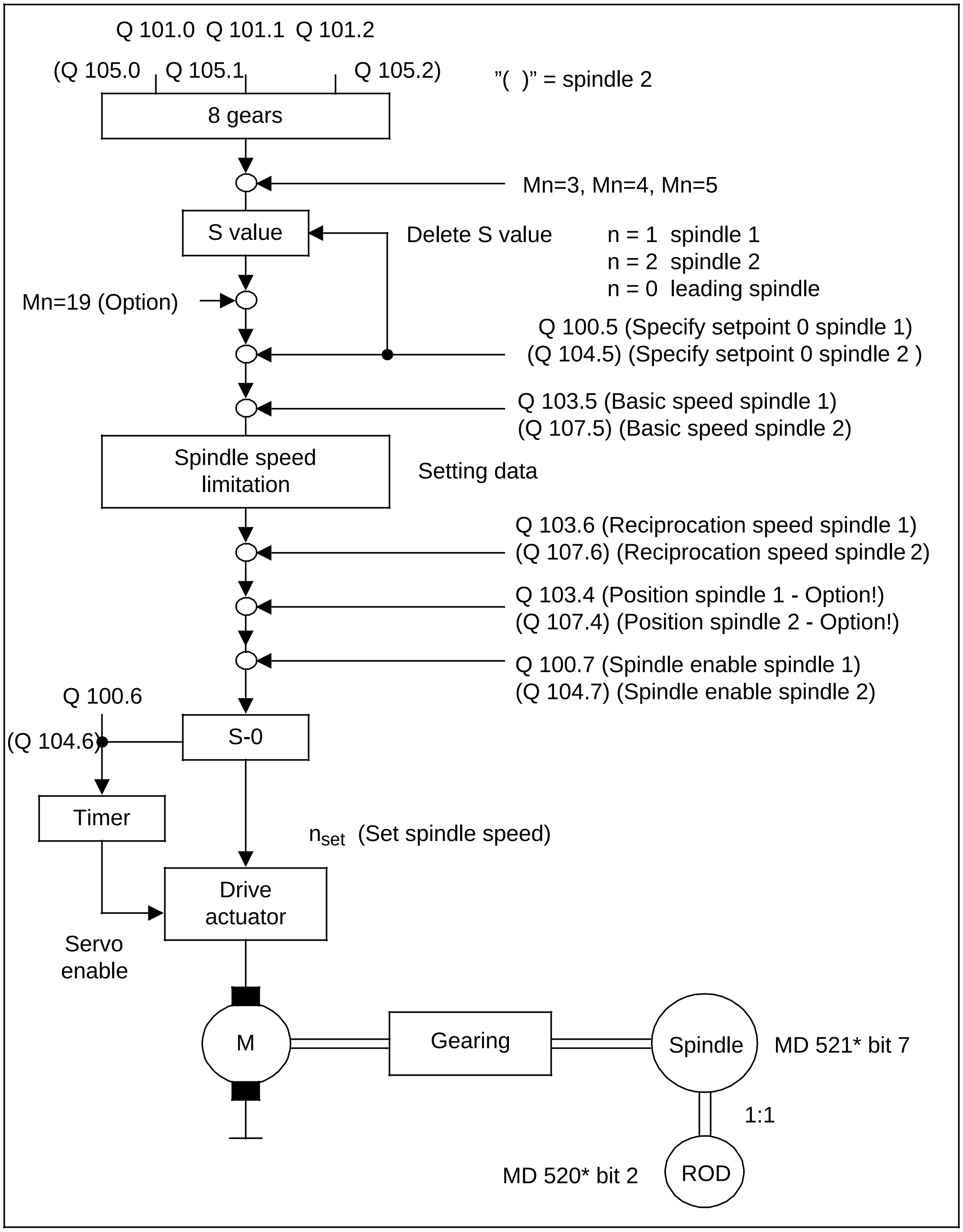
$$p = \frac{1.2}{200\ mV} \cdot 360° = 2.16°$$ positioning accuracy

Arithmetically speaking, an enhancement of positioning accuracy could be achieved by increasing the gain. However, it is important to remember that this also increases the spindle's tendency to oscillate.

10.1.4 Spindle influencing by PLC

The flowchart below is intended to show the effect of the individual PLC interface signals on the spindle. The feedback pulses are not shown for reasons of clarity.

Automatic gear selection (Q 101.3 or Q 105.3) is described in detail in Interface Description Part 1.



10.2 Approach to reference point

10.2.1 Corresponding MDs

- MD 240* (Reference point value)
- MD 244* (Reference point shift)
- MD 284* (Reference point cutoff speed)
- MD 296* (Reference point approach speed)
- MD 5008 bit 5 (Setting up in jog mode)
- MD 560* bit 6 (Approach to reference point with automatic identification of direction)
- MD 564* bit 0 (Direction of approach to reference point)

Indirectly related:

- MD 5004 bit 3 (NC START without reference point)
- MD 560* bit 4 (No reference point start inhibit)

10.2.2 Identification of direction with approach to reference point

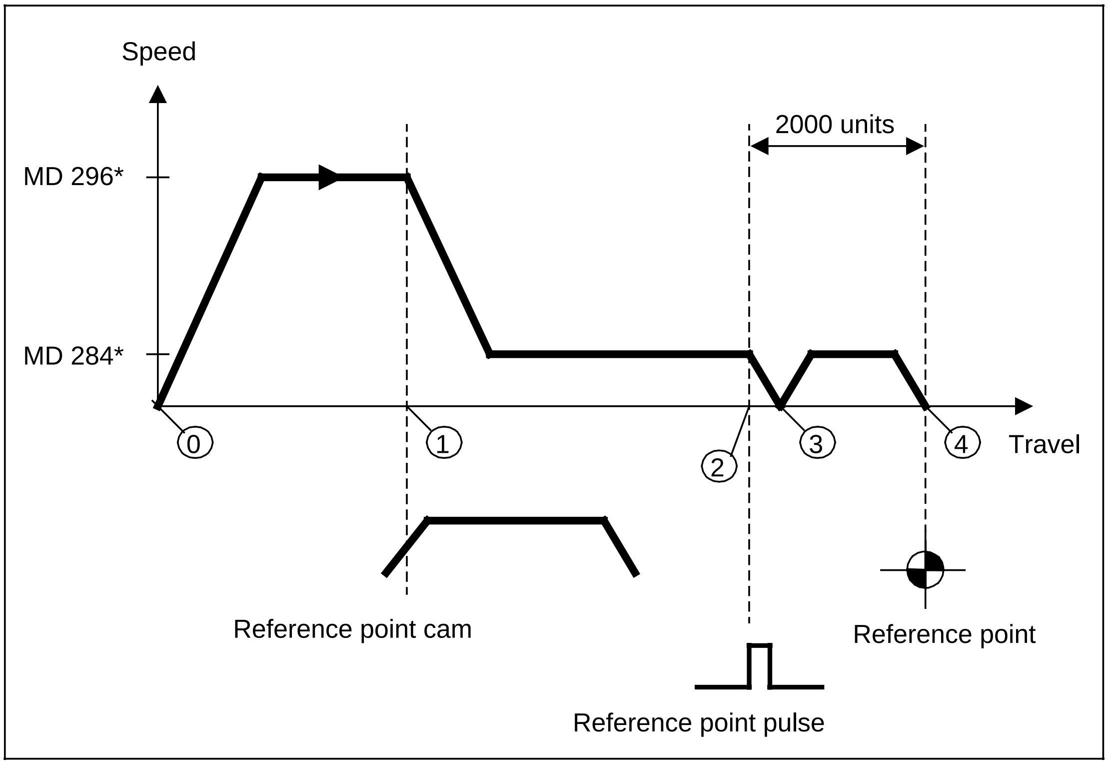
The control makes it possible to approach the reference point in two different ways - with or without automatic identification of direction. Selection is performed via MD 560* bit 6.

10.2.2.1 Approach to reference point without automatic identification of direction

Prerequisites:

- MD 560* bit 6 = 0
- Axis-specific feed enable is set
- Common feed enable is set
- Reference point is located between reference point cam and limit switch

Case 1: Axis ahead of reference point cam



When the correct direction key is actuated, approach to the reference point for the axis concerned is initiated in the specified direction (MD 564* bit 0) at the speed in MD 296*.

When the reference point cam is reached, the axis speed is reduced to the value in MD 284* via the ”Deceleration” interface signal.

After the reference point cam has been left, the next reference point pulse is evaluated and the axis braked.

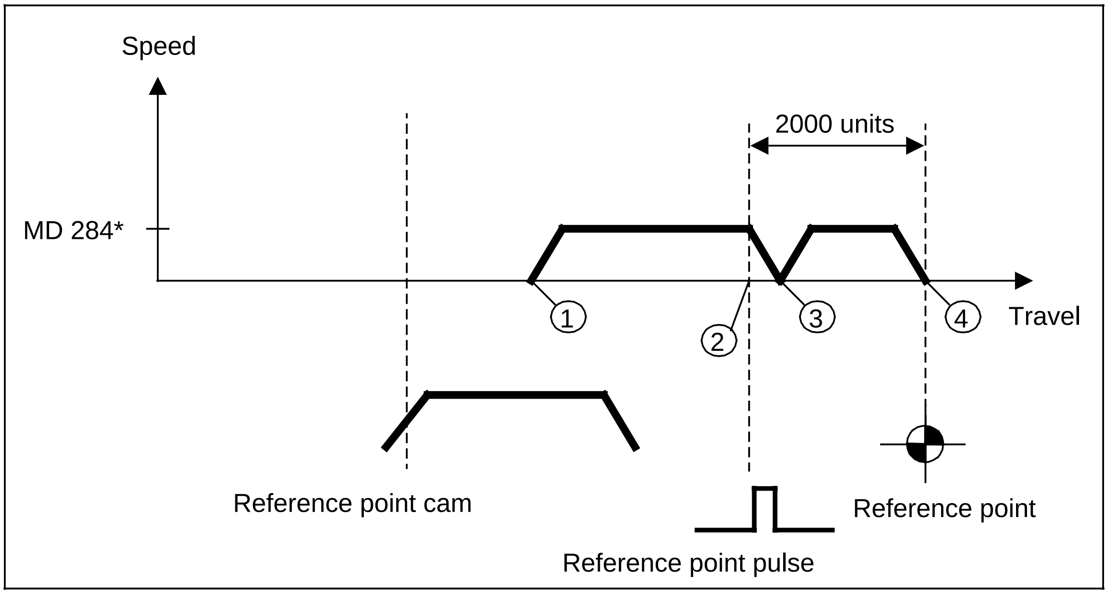
In order to prevent machine backlash during approach to the reference point, a fixed distance of 2000 units is covered from the reference point pulse to the actual reference point.

In view of the fact that point is at different locations for different speeds, the distance to go ( ) must be determined before approach to the actual reference point. To this end, the axis brakes to zero speed.

Reference point reached.

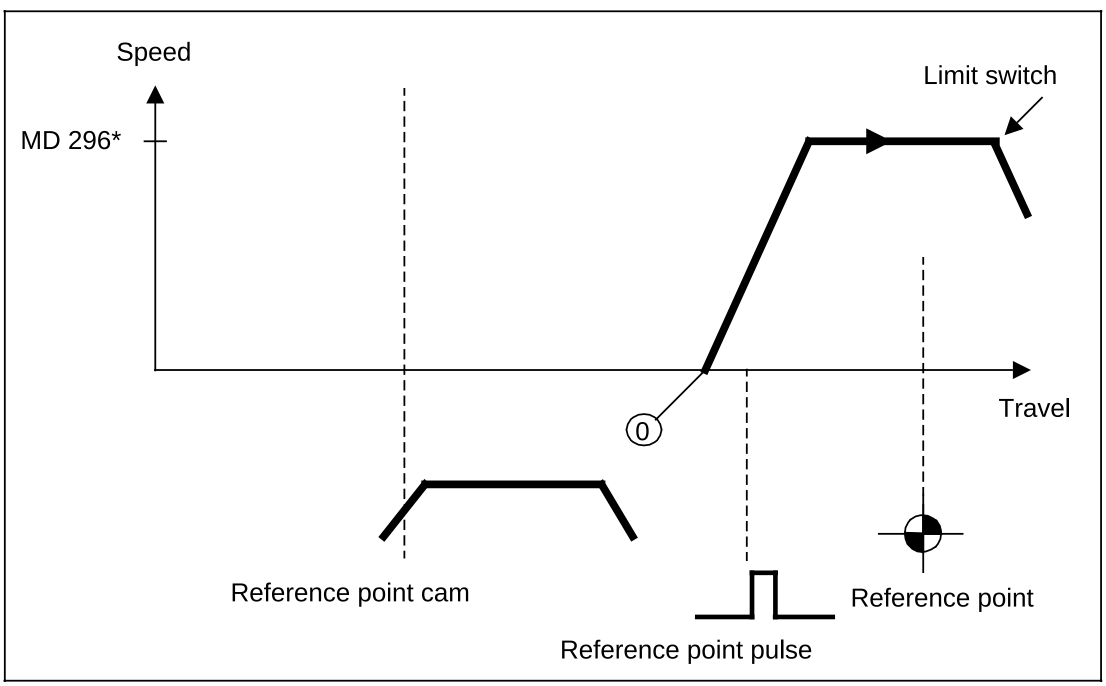
Case 2: Axis on reference point cam
Reference point approach speed

The axis accelerates immediately to the reference cutoff speed (MD 284*) instead of the reference point approach speed.



Case 3: Axis behind reference point cam

Since the condition of the "Deceleration" signal behind the reference point is the same as for the signal in front of this point, the control assumes that the axis is ahead of the reference point cam and accelerates to the reference point approach speed (MD 296*), i.e. in Case 3 it moves at high speed to the limit switch (EMERGENCY STOP) since the software limit switches prior to or during approach to the reference point are not active.



Complex travel interlocks had to be integrated into the PLC to obviate Case 3. It was thus decided to offer a facility which excludes Case 3 with approach to the reference point without any additional PLC support. This function is called approach to reference point with automatic identification of direction.

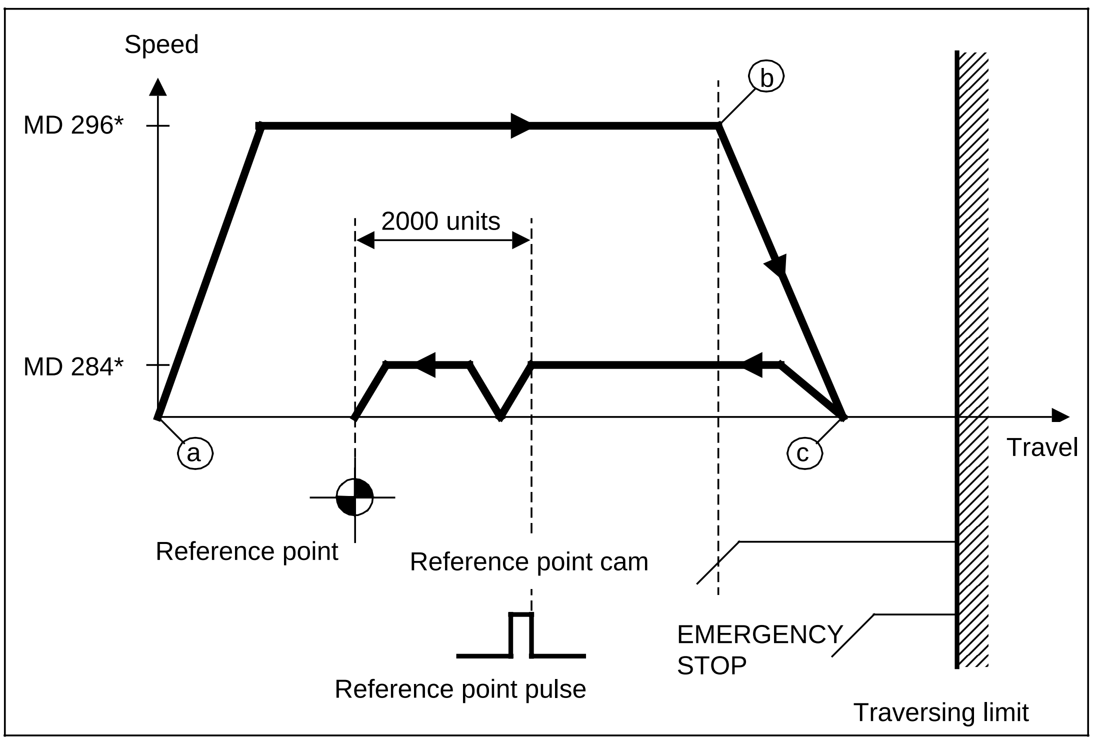
10.2.2.2 Approach to reference point with automatic identification of direction

Preconditions:

- MD 560* bit 6 = 1
- Feed enable signals set
- Reference point cam reaches as far as traversing limit
- Reference point in front of reference point cam

Automatic identification of direction is designed to prevent Case 3 during approach to the reference point without automatic identification of direction.

Case 1: Axis ahead of reference point cam

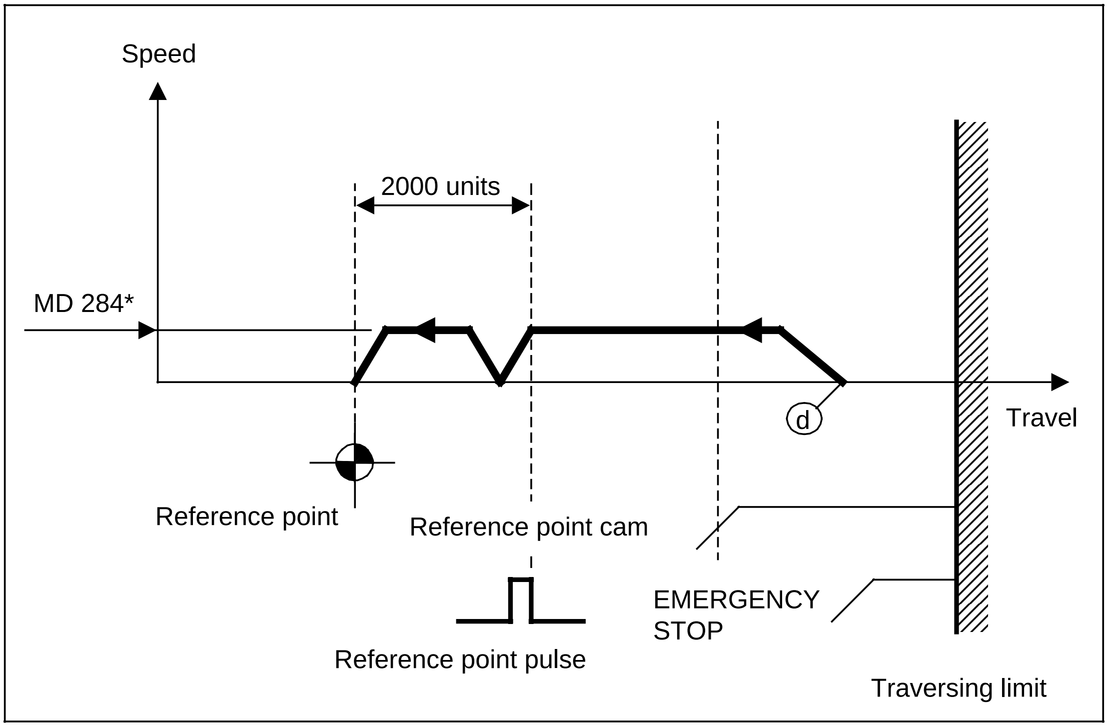


When the direction key is actuated, approach to the reference point for the axis is initiated in the specified direction (MD 564* bit 0) at the speed in MD 296*.

When the reference point cam is reached, the axis is braked to zero speed with the ”Deceleration” signal.

Deceleration from the reference point cam is performed at reverse speed (MD 284*) and the next reference pulse is evaluated (see Section 10.2.2.1 for a detailed description of the remainder of the sequence).

Case 2: Axis on reference point cam



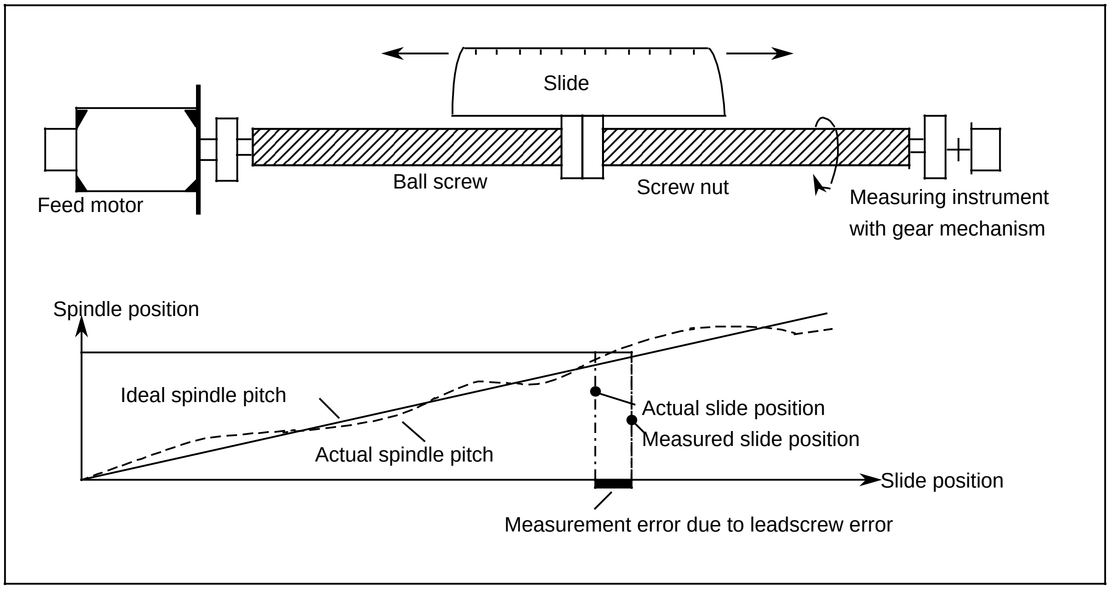
When the direction key is actuated, the CNC can determine precisely from the ”Deceleration” PLC signal that the axis is already on the reference point cam. The axis thus accelerates in the opposite direction (compared to MD 564* bit 0) to the speed in MD 284* (see Section 10.2.2.1 for a detailed description of the remainder of the sequence).

Case 3: Axis behind reference point cam

Case 3 cannot occur.

10.3 Leadscrew error compensation

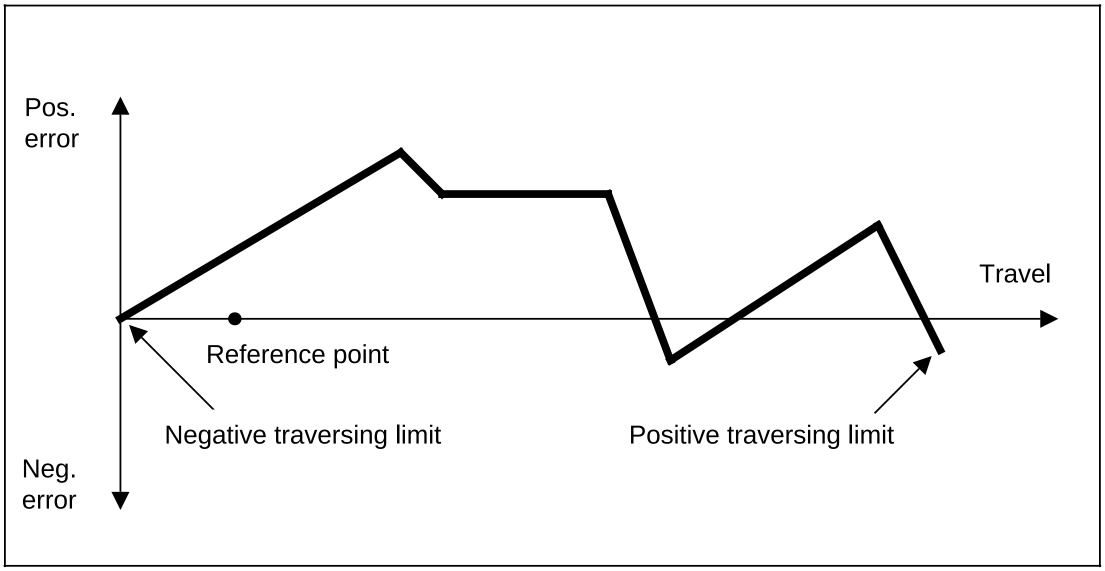
The principle of ”indirect measurement” in NC-controlled machines assumes that the pitch of the ball screw is constant at any point within the traversing range so that the actual axis position may be derived from the position of the drive spindle. However, varying deviations result from manufacturing tolerances in the various spindle qualities. In addition, measuring system errors (though comparatively insignificant) and any other possible machine-dependent errors must be taken into account. The sum check error may be determined by plotting an error curve over the entire traversing range of the axis. The reference measuring system used must be a high-precision instrument, e.g. a laser interferometer. The dimensional deviation at the workpiece can be significantly reduced as a result of appropriate compensating values which are input in the control at the installation stage.



Indirect measurement

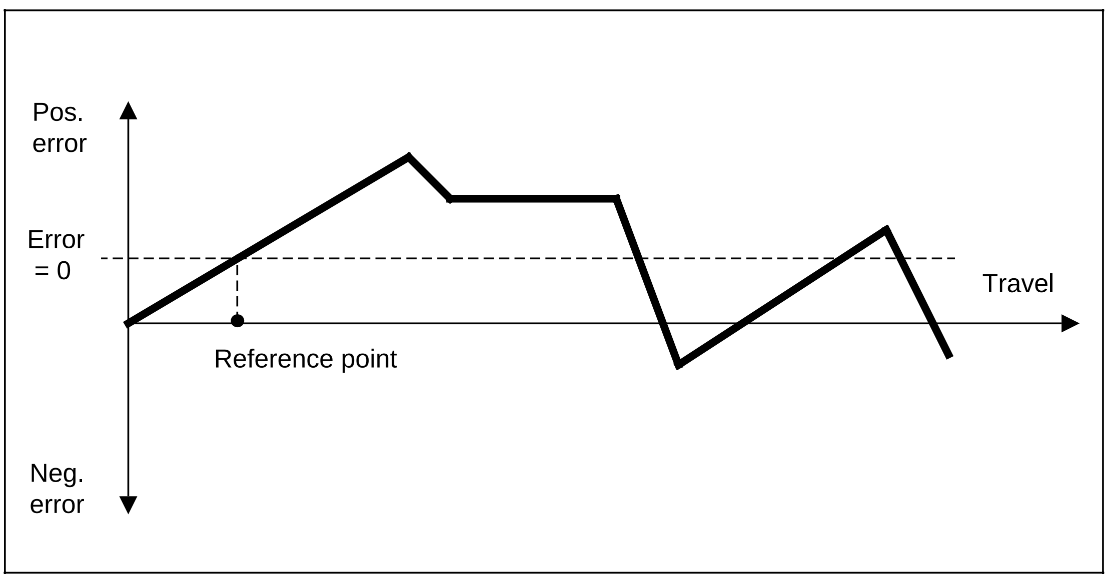
Measurement of leadscrew pitch of an axis

To synchronize the measuring system the reference point must first be approached. This is then followed by travel to the negative traverse range limit of the axis, commencing from this point an error curve is then plotted in the positive direction using an accurate instrument; the reference point must be identified.



Error curve recorded

In view of the fact that compensation is not possible at the reference point, the error curve must be shifted so that the error is zero at the reference point.



Error curve with error = 0 at reference point

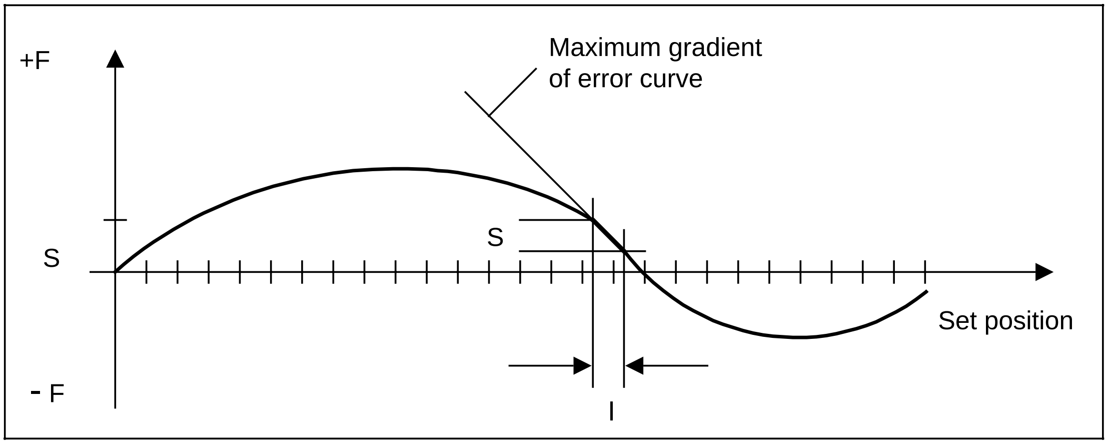
To compensate the leadscrew pitch in both directions a second error curve must be recorded from the positive to the negative traverse range limit and shifted so that the error is zero at the reference point.

Spacing between two compensation points, compensation amount

In SINUMERIK 810 GA3/820 GA3 the total of 1000 compensation points (input in MD 6000 to 6249) are available for all axes. The spacing between two compensation points depends on the tolerance of the error curve compensated, the actual lead screw error and the number of possible compensation points.

The following method for determining the spacing between 2 leadscrew error compensation points may be used:

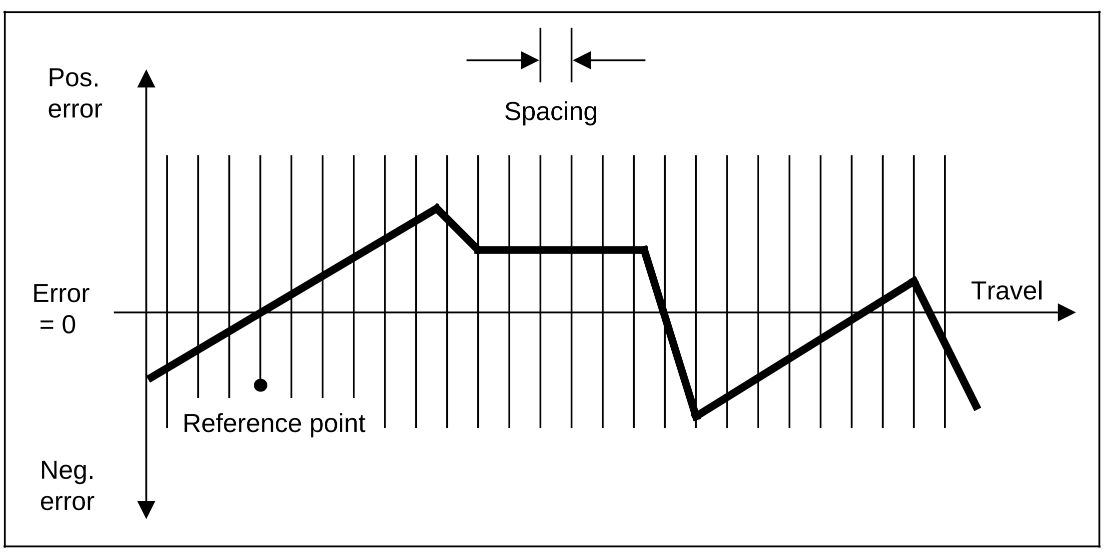
S	:	Compensation amount, e.g. 1/2 tolerance band	=	MD 328*
I	:	Spacing between 2 leadscrew error compensation points	=	MD 324*



Determination of leadscrew error between two points

The point with the greatest pitch error is determined and the travel (I) in which the specified compensation amount (S) is passed through is established.

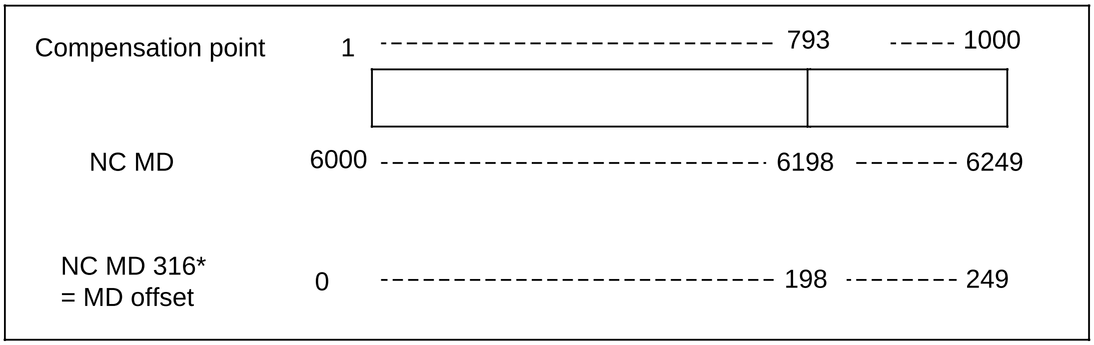
The relevant compensating value for the spacing is based on the permissible tolerance band and should be selected so that the compensated error curve approximates as closely as possible to the ideal condition.



Running the error line through the compensation point

Reference point - compensation point

It is then specified how many compensating points must be supplied by means of the entered spacing between 2 leadscrew error compensation points and the end stops at the machine. Since leadscrew error compensation is only active with synchronization of the axis - at the reference point - particular significance is attached to the compensation point coinciding with the reference point. This compensation point is entered in encoded form in MD 316*. The compensating value at this point must be 0.



Determining the compensation point as a machine data value (MD)

As the compensation point is not entered in MD 316* directly but the MD offset (6125: MD offset = 125) the reference point can only lie on the compensation points 1, 5, 9, 13, 17... .

<table>
<tr><th>MD No.</th><th>Bit 7</th><th>6</th><th>5</th><th>4</th><th>3</th><th>2</th><th>1</th><th>0</th></tr>
<tr><td>6000</td><td colspan="2">Comp. point 4
yes / no + / –</td><td colspan="2">Comp. point 3
yes / no + / –</td><td colspan="2">Comp. point 2
yes / no + / –</td><td colspan="2">Comp. point 1
yes / no + / –</td></tr>
<tr><td>6001</td><td colspan="2">Comp. point 8
yes / no + / –</td><td colspan="2">Comp. point 7
yes / no + / –</td><td colspan="2">Comp. point 6
yes / no + / –</td><td colspan="2">Comp. point 5
yes / no + / –</td></tr>
<tr><td>6002</td><td colspan="2">Comp. point 12
yes / no + / –</td><td colspan="2">Comp. point 11
yes / no + / –</td><td colspan="2">Comp. point 10
yes / no + / –</td><td colspan="2">Comp. point 9
yes / no + / –</td></tr>
<tr><td>– / / –</td><td colspan="6" rowspan="2"><table><tr><th>yes/no</th><th>+/-</th><th></th></tr><tr><td>1</td><td>1</td><td>pos. comp.</td></tr><tr><td>1</td><td>0</td><td>neg. com.</td></tr><tr><td>0</td><td>irrelev.</td><td>not comp.</td></tr></table></td><td colspan="2" rowspan="2"></td></tr>
<tr><td>– / / –</td></tr>
<tr><td>6428</td><td colspan="2">Comp. point 996
yes / no + / –</td><td colspan="2">Comp. point 995
yes / no + / –</td><td colspan="2">Comp. point 994
yes / no + / –</td><td colspan="2">Comp. point 993
yes / no + / –</td></tr>
<tr><td>6249</td><td colspan="2">Comp. point 1000
yes / no + / –</td><td colspan="2">Comp. point 999
yes / no + / –</td><td colspan="2">Comp. point 998
yes / no + / –</td><td colspan="2">Comp. point 997
yes / no + / –</td></tr>
</table>

Since 4 compensation points are available for each machine data, it is specified in the control that only the point on the far right-hand side (bits 0, 1) can be defined as the reference point.

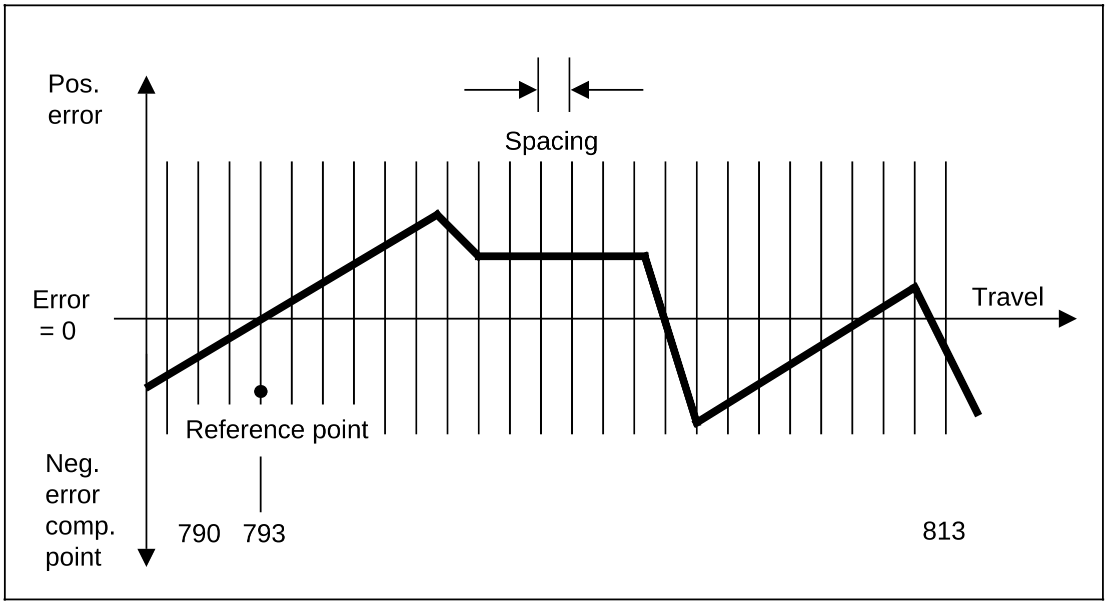
Example:

Reference point - compensation point 793 =n x 4+1

MD offset= $\frac{793 - 1}{4}$ =198= $\left(\frac{\text{comp.point} - 1}{4} \right)$

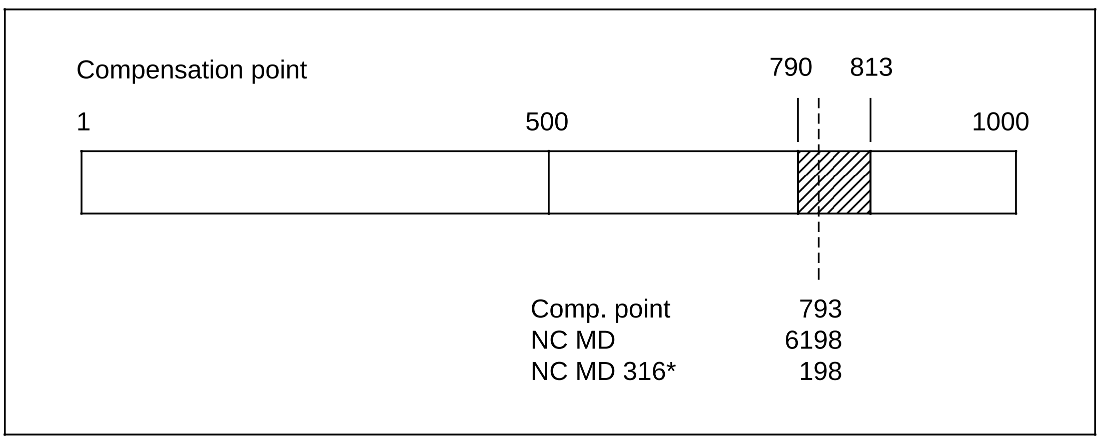
MD 316* has the value 198.

As stated above, the reference point determines which of the possible 1000 compensation points are used for the axis concerned.



Determining the compensation point using the reference point

If the reference point is assigned to compensation point 793, breakdown of the 1000 compensation points is as follows:



Determining the compensation point as a machine data value

As can clearly be seen, the reference point has determined the location of the hatched area of the compensation points used. This area terminates at point 790 or 813 due to the spacing between the leadscrew error compensation points and the maximum traversing range of the axis.

If leadscrew error compensation is used for a number of axes, the person responsible for installation must ensure when inputting the MDs that the compensation points do not overlap during traversing as no check is carried out in the control.
However, the gaps between the axes can be of any size, provided the overall range of 1000 compensation points is not exceeded.

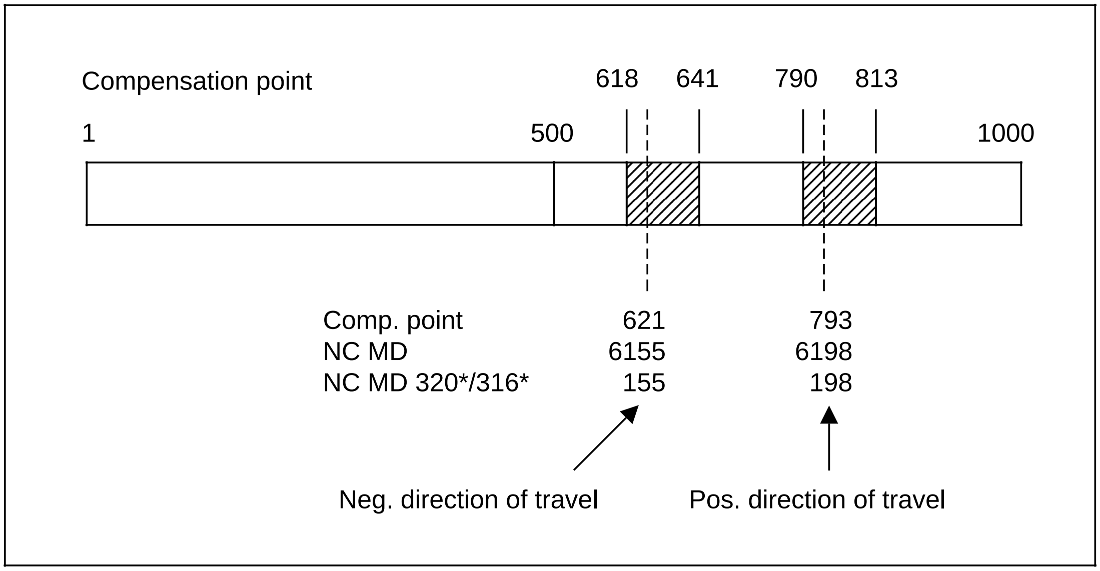
Direction-dependent leadscrew error compensation

For this an error curve must be recorded in the positive to negative direction.

In the case of ball screws, prestressing of the screw nut yields an identical error curve, irrespective of the plot direction during measurement. However, with worm drives, significant differences may arise between the positive and negative directions of travel. Consequently, an error curve must also be plotted in the negative direction and input as compensation.

The procedure is similar to that for the positive traversing movement, ensuring that the compensation ranges do not overlap between the positive and negative traversing movements and between the axes. Since the reference point again determines with this compensation curve where the compensation points lie within the 1000 points, the reference point must be entered in NC MD 320* in encoded form (MD offset).

Example:



Determining the compensation point as a machine data value

Machine data for leadscrew error compensation:

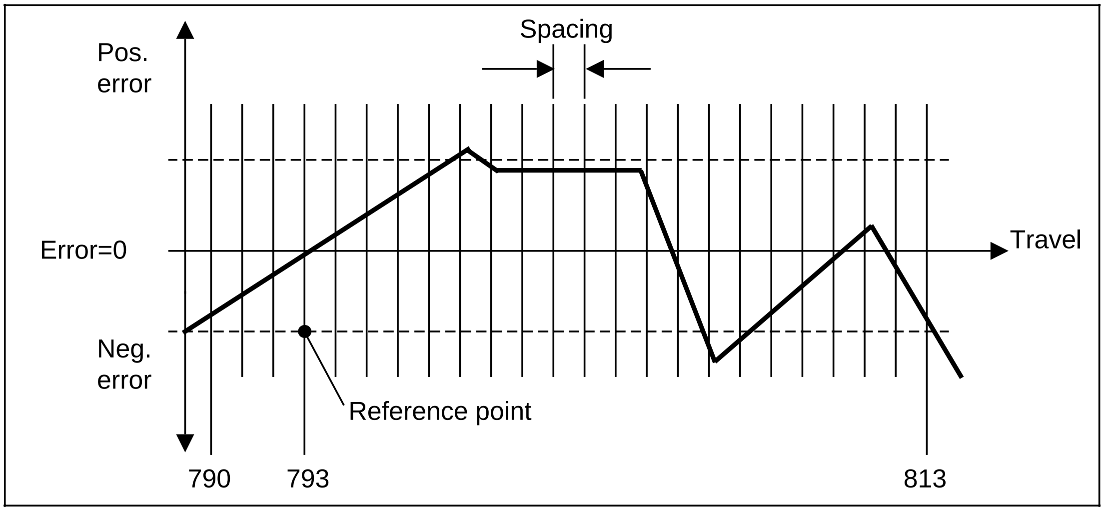
- MD 316* (Positive compensation pointer)
- MD 320* (Negative compensation pointer)
- MD 324* (Spacing between 2 leadscrew error compensation points)
- MD 328* (Compensating value)
- MD 6000 - 6239 (Compensation points)

Any modification to the MDs is only active after Power On and reference point approach. Since the compensating value at the compensation point must be processed as quickly as possible, the input acceleration (NC MD 276*) is not applicable in this case. Consequently, the compensating value (NC MD 328*) is limited to max. 100 units.

Note:

Both direction-dependent and direction-independent leadscrew error compensations are options and must therefore be ordered with the order code H56.

Example:



Determining the compensation point using the reference point

Axis 1 displays the error curve shown in the diagram. So far no compensation points have been used.

Reference point value 0
Max. traverse path in negative direction - 35.000 mm
Max. traverse path in positive direction 205.00 mm

Tolerance band (is defined by the machine manufacturer) e.g. 0.01 mm. Define spacing between two leadscrew error compensation points, e.g. 10 mm:

Defining the number of compensation points

Left from the reference point (A-) 35 mm traverse distance/max-

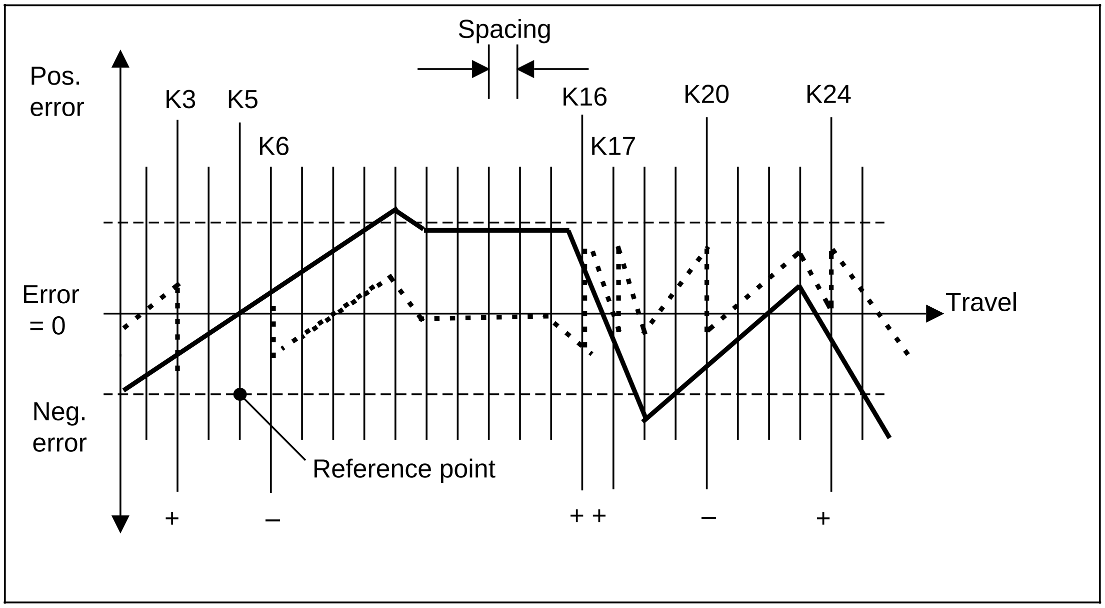
A – At 10 mm great spacing this results in **3 compensation values**

Right from the reference point (A+) 205 mm traverse distance/max+

A+ At 10 mm great spacing this results in **20 compensation values**

Total number of compensation values:
K =(A-) +(A+) + (Ref) = 3 + 20 + 1 **24 compensation values**

This results in NC MD 3161 = 1 (NC MD 6001), e.g. compensation point K5 is the point at which the reference point is located; here no compensation can be made. By traversing 10 mm in the negative direction compensation point K4 is used for compensation. By traversing 10 mm in the positive direction compensation point K6 is used.



The new error curve using the determined compensation points

- - -	Tolerance band e.g.	10 µm
- - - -	Compensation value e.g.	5 µm
▪ ▪ ▪ ▪ ▪	Compensated curve	

Departing from the reference point in the negative direction the error curve runs up to the end of the traverse path within the tolerance band. It does not have to be compensated. A better result is obtained if positive compensation is made at K3. To remain as close as possible to error 0, it is necessary to compensate in the positive direction negatively at K6, positively at K16 and K17, negatively at K20 and positively again at K24. The new error curve would then be the one shown in the diagram as a dotted line.

The following machine data would have to be set:

- Option leadscrew error compensation
- NC MD 3161 = 1 (this defines the reference point , K5)
- NC MD 3241 = 10000 (grid spacing 10 mm)
- NC MD 3281 = 5 (compensation value 5 µm)
- NC MD 6000 = 00 11 00 00 (at K3 positive compensation)
- NC MD 6001 = 00 00 10 00 (at K6 negative compensation, bits 0 and 1 must be 0)
- NC MD 6002 = 0 no compensation
- NC MD 6003 = 11 00 00 00 (at K16 positive compensation)
- NC MD 6004 = 10 00 00 11 (at K17 positive comp. and at K20 negative compensation)
- NC MD 6005 = 11 00 00 00 (at K24 positive compensation)

10.4 Rotary axis function

10.4.1 Corresponding data

All data as for linear axes but with the following additions

• NC MD 344*	(modular value rotary axis for leadscrew error compensation)	
• NC MD 560* bit 7	(actual value display 360°)	
• NC MD 560* bit 3	(rounding for rotary axes)	also active for linear axes
• NC MD 560* bit 2	(rounding whole/half degree)	also active for linear axes
• NC MD 564* bit 5	(position control for rotary axis)	
• NC MD 572* bit 2	(rotary axis Modulo 360° - programming)	
• NC-MD 572* bit 4	(full/half circle for rotary axis)	
• Alarm 100*	(grid spacing not permissible leadscrew error compensation)	
• Alarm 2064	(program error with rotary axis)	

10.4.2 Function description

Different demands are made on a rotary axis depending on the type of machine. The rotary axis function is, therefore, divided into three subfunctions, which are activated via machine data or programming. The control can be adapted to the various machine types by combining subfunctions.

„Rotary axis”: NC MD 564* Bit 5

With this machine data the axis is defined as a rotary axis. The display is absolute (1 rev. 360°, 2 revs 720° etc.), and the @ functions, too. However, the axis is programmed like a linear axis. The units of the axis-specific NC MDs are treated differently.

Unit 10^{-3} degrees for position control resolution 1/2 x 10^{-3} units and input resolution 10^{-3} units.

Actual value display „Modulo 360°”: NC MD 560* Bit 7

When the bit is set, the display is Modulo, i. e. the display is reset to 0 after 359.999 degrees. The axis is programmed like a linear axis.

„Modulo programming”: NC MD 572* Bit2

By setting this bit the axis can now be programmed as a rotary axis.

The first block to be traversed of the rotary axis in a part program is always traversed on the shortest path. Using the G68 function it is always possible to approach the block final value on the shortest path.

G68 is modal and belongs to the G90/91 group. If ”Modulo programming” is not activated, G68 is treated like G90.

If the rotary axis is not to traverse the shortest path, it must be programmed with G90 and a sign.

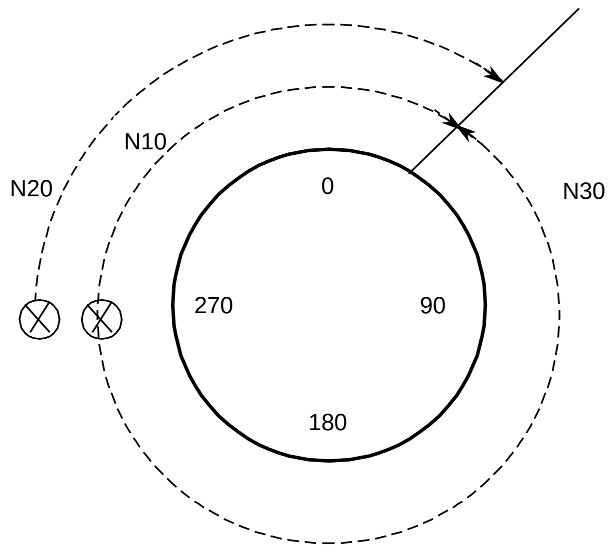
Example:

Axis is at 270°

1) N10 G68 C10
   Traversing on the shortest path

2) N20 G90 C10
   Traversing in the positive direction
   to the end point

3) N30 G90 C-10
   Traversing in the negative direction
   to the end point



The machine data ”Modulo 360” and ”Modulo programming” are only permissible with the machine data ”Rotary axis”.

Combination of subfunctions:

Rotary axis	Modulo program	Modulo 360°	Significance
0	0	0	Linear axis
1 1 1	0 0 1	0 1 1	Application permitted
1 0 0 0	1 0 1 1	0 1 1 0	Application illegal

Axis-specific machine data

If an axis is defined as a rotary axis (NC MD 564* Bit 5), the following applies depending on NC MD 5002 (input resolution and position control resolution):

a) 1 unit = 2 position control resolutions (reference system MS)
e.g. 1 position control resolution= 0.5 x 10^{-3} degrees
1 unit = 10^{-3} degrees

b) 1 unit = 1 unit = 1 input resolution (reference system IS)
e. g. 1 input resolution= 10^{-3} degrees
1 unit = 10^{-3} degrees

All axis-specific machine data are specified in degrees for a rotary axis,
e.g.: Maximum speed of a rotary axis= 15 revolutions/min

15 rev/min x 360 degrees/rev = 5400 degrees/min
Input in NC MD 280* = 5400 (1000 degrees/min)

10.5 On-the-fly synchronization of rotary axis (option)

For the functions

- TRANSMIT for turning machines
- ”Interpolatory tapping without compensating chuck” for milling machines
- ”Interpolatory thread cutting” for turning machines, and
- ”Cylindrical interpolation” for turning machines

it is necessary to constantly change over from spindle mode to rotary axis mode and vice versa. With the function ”On-the-fly synchronization of rotary axis”, the user is provided with the means for rapidly changing over and synchronizing. After changing mode, it is not necessary to perform a reference point approach for the rotary axis provided there is no PARKING AXIS signal. It is not possible to combine this function with other axis changeovers.

The changeover between spindle and rotary axis can take place continually once the spindle has been synchronized.

The following additional NC machine data must be observed:

MD 260	M function number for selecting the rotary axis
MD 261	M function number for deselecting the rotary axis
MD 461*	Assignment of rotary axis to spindle
MD 521* Bit 1	Rotary axis remains selected even after RESET.

For controlling the rotary axis selection, PLC output Q 101.7 (for spindle 1) or Q 105.7 (for spindle 2) is required.

In the machine data MD 260 and MD 261, the M function numbers must be defined for changeover. The extended address notation is used in programming:

M1 = <MD 260> or <MD 261> for spindle 1
M2 = <MD 260> or <MD 261> for spindle 2

If the spindle is active, then the associated rotary axis is not available. The value 0 is then entered in the position display for this rotary axis. When changing over from spindle to rotary axis mode, the rotary axis has already been referenced (exception: when the PLC signal ”park” is active).

10.5.1 Allocation of rotary axis to spindle

A rotary axis can be allocated to spindle 1 (2) via machine data MD 4610 (4811).

Example:

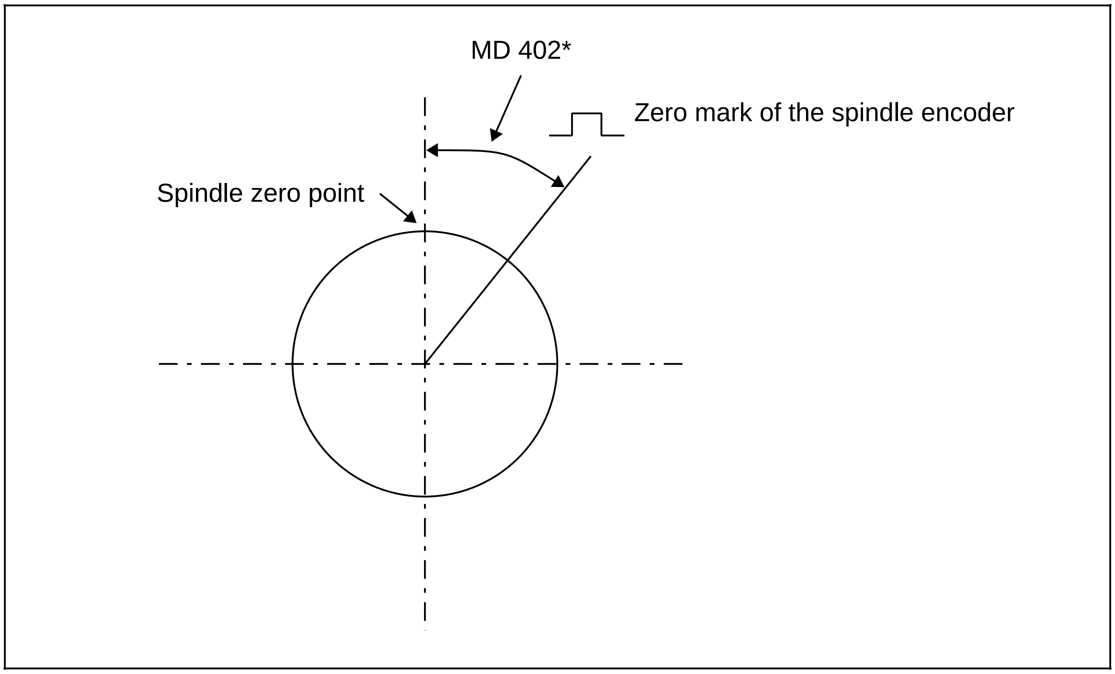
MD 4610 = 3	MD 4611 = 4
3rd axis	4th axis
Spindle 1	Spindle 2

A spindle cannot be allocated to a rotary axis when

1. the limiting value for number of axes has been exceeded
2. the axis does not exist or is a simulation axis
3. no setpoint value has been assigned to the spindle in MD 400* and no pulse weighting through MD 520*, bit 3. Alarm 8 ”Wrong axis/spindle allocation” is output.
4. the axis is not a rotary axis (MD 564*, bit 5=0). The alarm 2183 ”axis is not a rotary axis” is output. The alarms lock NC START and NCREADY2.

10.5.2 Reference points for the spindle and for the associated rotary axis

The zero point for the spindle results from the zero mark of the spindle encoder and the zero mark offset (MD 402*)



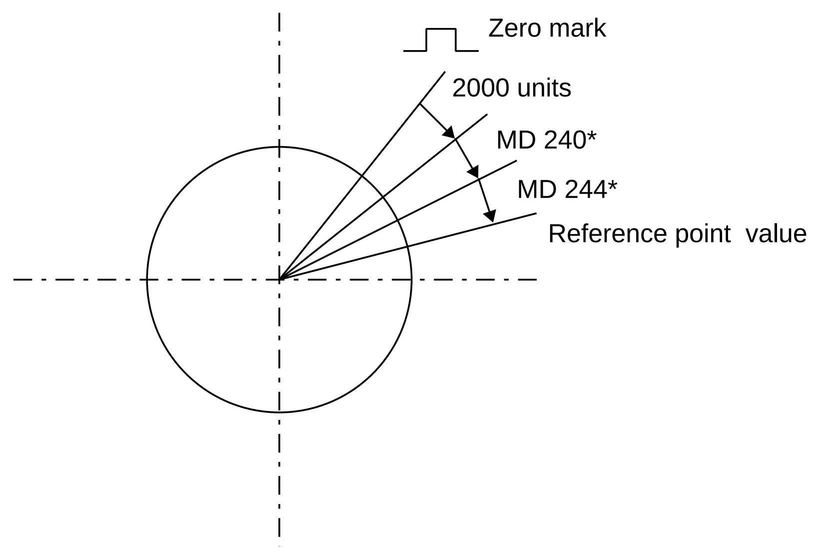
The following then applies for the zero point of the associated rotary axis:

- where there is a common encoder for spindle and rotary axis, from the position of the zero mark of the encoder

  + reference point value (MD 240*)
  + reference point offset (MD 244*)
  + 2000 units (see Section 10.2.2)

- where there are separate encoders, from the zero mark of the rotary axis encoder

  + reference point value (MD 240*)
  + reference point offset (MD 244*)
  + 2000 units (see Section 10.2.2)



10.5.3 Selection and deselection of rotary axis mode

Selection of rotary axis mode

Selection is by means of the extended M function given in MD 260. The extension of the M function specifies the spindle. Selection can take place through the part program or by overstoring the M function.

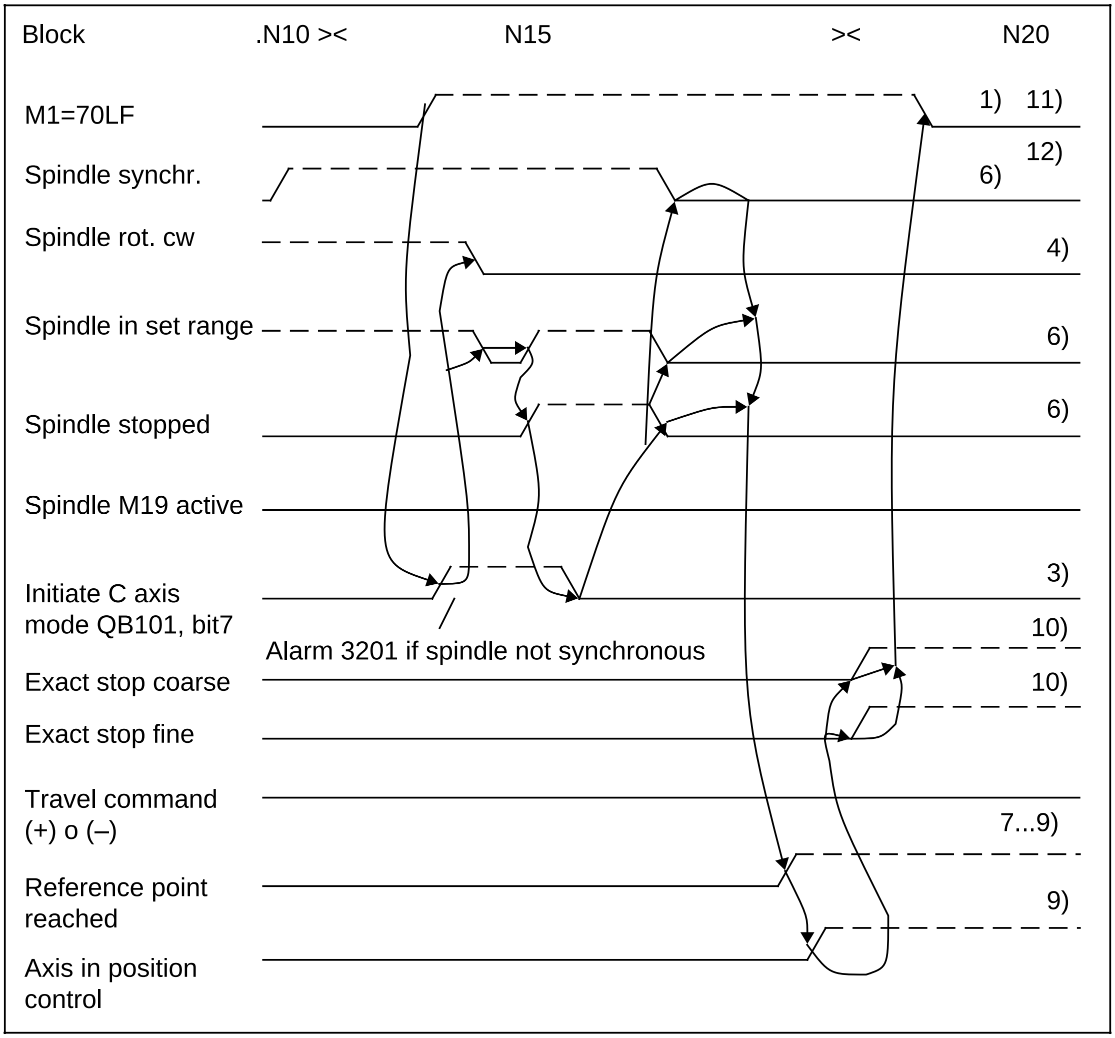
Prerequisite for selection:

- The rotary axis has been allocated to the spindle
- The axis selection must be alone in the block
- The PLC signal PARKING AXIS is not on
- M19 must not be active.

Note:
For this function, no encoder with more than 16383 pulses may be used (number of pulses=number of encoder pulses x 4, and this must be less than 64k)

Program example (MD 260 = 70, spindle 1)

N10...LF
N15 M1=70 LF
N20...LF



1) M1=70 reaches the PLC interface.
2) The user can stop the spindle and change gear stages (not shown in the example).
3) The user applies the signal ”Initiates C axis mode” to the spindle interface
4) If the spindle has not been stopped by the user, the NC stops the spindle.
5) The user could change gear stages.
6) The spindle is deactivated.
7) The axial input signals are read.
8) The C axis is still in follow-up mode.
9) The position controller is closed.
10) The system waits for exact stop coarse/fine.
11) Block preparation is synchronized.
12) The next block is executed.

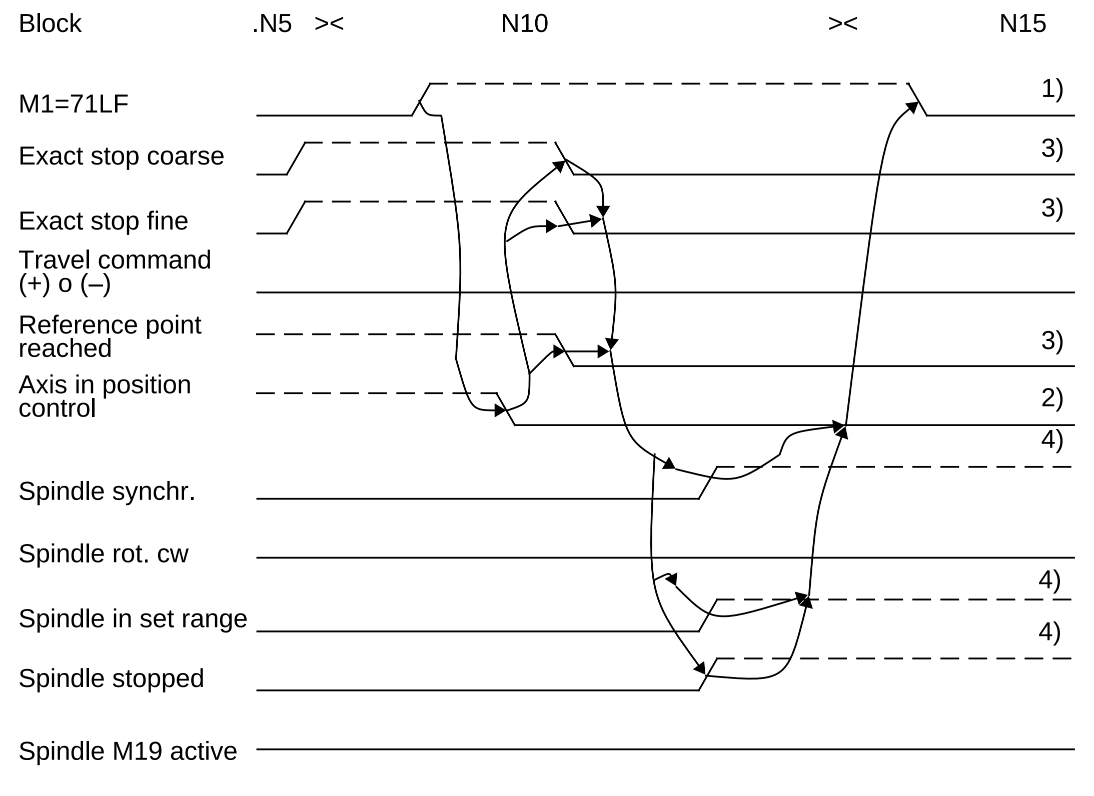
Notes:

- If the spindle is not synchronized, then alarm 3201 is issued if QB 101 bit 7=1. The spindle must then be synchronized by rotating it again (or the alarm must be acknowledged with the cancel key).
- If rotary axis selection is programmed when M19 is active, alarm 3202 is output.
- For the second spindle, the sequence is the same as for rotary axis selection. Q 105.7 is used as PLC bit for the second spindle.

Deselection of rotary axis mode

Program example (MD 261 = 71, spindle 1)

N05...LF
N10 M1=71LF
N15...LF



A M1=71 block has been executed when the spindle signal s are on.

1) M1=71 reaches the PLC interface and the NC.
2) The C axis is switched to follow-up mode.
3) The axial input signals of the PLC are no longer set by the NC. The axial output signals are no longer considered.
4) The spindle is activated. The spindle-specific signals of the PLC are set by the NC.
5) The next block can be executed.

The rotary axis mode must be deselected in the same channel as it was selected.

Note:

If the PARKING AXIS signal has been activated for the rotary axis, the spindle is no longer synchronized after deselection of the rotary axis, i.e. the spindle must have rotated by at least one complete revolution before the next positioning operation.

10.5.4 Block search

The operator must overstore to select/deselect the rotary axis mode before block search takes place.

10.5.5 Behaviour in spindle mode

- For the rotary axis, no axial input signals are sent to the PLC and no output signals are read by the PLC.
- The position display for the rotary axis in the basic displays and in the SERVICE AXIS DATA is on zero.
- MD 5004 bit 3 and MD 560* bit 4 (NC start interlock) have no meaning for the rotary axis.
- Programming of the rotary axis leads to NC stop (alarm 1961).
- The rotary axis cannot be traversed in the JOG, INC, REPOS, REF modes (alarm 1961).

10.5.6 Behaviour in rotary axis mode

- The positional value can be between 0 and 360 degrees.
- The set spindle speed is deleted.
- The spindle cannot be programmed. If an attempt is made to do so, the control issues alarm 3000.
- The spindle cannot be overstored.
- The rotary axis can be referenced via REF or G74.
- The spindle-specific input and output signals are ignored by the PLC.

10.5.7 Special features

- Selection and deselection of the rotary axis mode can also be performed in simulation. If the rotary axis mode is selected by M1=17, then the position display is on zero in simulation.
- By means of MD 521* bit 2, the user defines the behaviour of the rotary axis mode at RESET.

10.6 Interpolatory tapping/thread cutting without compensating chuck (G36)

This function is an option.

10.6.1 Description of function

Prerequisite:

Rotary axis mode for the spindle concerned.

The function G36 creates a linear interpolation between the spindle in rotary axis mode and the infeed axis. The second KV factor is selected for the interpolation when G36 is selected. If the second KV factor has the value zero, the first KV factor continues to be used.

With G36, face/longitudinal threads (spindle in rotary axis mode and one linear axis) and taper threads (spindle in rotary axis mode and two linear axes) can be processed. Threads with increasing and decreasing lead cannot be produced.

The maximum achievable rotary axis speed is 13638 rev/min (with the standard setting of MD 155) for a positional control resolution of $0.5 \cdot 10^{-1}$ degrees. If the input resolution is set to $0.5 \cdot 10^{-3}$ then a maximum of 2982 revolutions can be travelled in each G36 block. The ”Modulo 360 degree programming” function must not be selected for the rotary axis (MD 572* bit 2, problems with block search). The important point is that the axes involved in interpolatory tapping have approximately the same dynamic response.

Additional signals:

I 103.4	PLC input signal ”G36 active” Channel 1
I 109.4	PLC input signal ”G36 active” Channel 2

Function G36 requires feed with function G98 (rotational feed for rotary axis in rev/min). Function G36 is a member of G group 0, i.e. deselection of the G36 function is achieved by programming another G function in G group 0 (e.g. G00).

10.6.2 General conditions

- All machine data for the axes apply
- No other feed type (except for G98) may be selected in a G36-active program part.
- G36 is active until deselected (modal)
- G 98 F... Feedrate for rotary axis in rev/min and it is contained in G group 11 (display in rev/min [rev]
- Precisely one rotary axis must be written for each G98-active traversing block.
- The traversing path must not be 0.
- G36 automatically ensures a block speed of 0. This means that, technologically speaken, no dwell is required in the reversal point.

- G98 can also be used without G36. In this case, the block speed stored in MD 3 is interpreted as linear feed and displayed as revolutional feed (rev/min).
- In the case of dry run feed, the feed stored in the setting data is taken and displayed as linear feed.
- G64 and G62 have no effect in G36-active blocks.
- The infeed axis must be assigned an interpolation parameter for G36 in MD 304*. This interpolation parameter must be written in the program. If this is omitted in the active G36 block, alarm 3006 is issued.
- Precisely one rotary axis and one linear axis must be programmed for cylindrical threads or one rotary axis and two linear axes for taper threads. In the case of taper threads, the lead parameter is assigned to the linear axis with the greater path. This means that taper threads with a slope of up to 45 degrees are possible.
- The ”Feedforward control factor”, ”Dead time constant” parameters have no effect with G36.
- When the G36 function has been deselected, the originally selected G function for G group 11 again becomes active.
- In each G36-active block, the rotary axis must be written. A numerical value behind the axis designation (e.g. C30) is ignored.
- The end value for the infeed axis and the thread lead are included in the program.

Example:

N30 G36 C Z – 30 K5 F100 L_F

rev/min
Thread lead
Infeed axis
Rotary axis

10.6.3 Diagnostics/start-up

When G36 is active, the display for the ”Contour deviation” in the ”Service axis” display is switched over and for the linear axis (lead axis) it shows the deviation of the follow-on path (in the input resolution) with respect to the rotary axis. By altering the K_V or MULTGAIN factor, it is thus possible to balance out the follow-on path difference to zero and to increase the dimensional accuracy. The axis which is dynamically slower can be identified by the sign: ”-” means that the K_V factor for the infeed axis is too large compared with the K_V factor for the rotary axis.

10.6.4 Display

The G functions G36 and G98 are displayed in the basic displays in the ”G functions” field.

The feed is displayed in rev/min (referred to the rotary axis) when G98 is active.

The feed type ”Revolutions per minute” is abbreviated to ”U” in the feed display (as an alternative to ”R” and ”M”).

10.7 Different input, position control and display resolutions

10.7.1 General

SINUMERIK 810 GA3/820 GA3 allows different input, position control and display resolutions to be defined for linear and rotary axes.

- Input resolution
  This is defined for all axes together in MD 5002 bits 5-7. The dimensional unit metric/inch is defined by MD 5002 bit 4.
- Position control resolution
  This is defined for a specific axis in MD 584* bits 1-3. The dimensional unit metric/inch is defined for all axes together by MD 5002 bit 0.
- Display resolution
  This is defined for a specific axis in MD 584* bits 5-7. The dimensional unit is defined like the input resolution by MD 5002 bit 4.

The velocities resulting from the resolutions and the possible combinations of the resolutions are shown below for linear and rotary axes.

10.7.2 Combinations with linear axes

10.7.2.1 Maximum velocities

The maximum axis velocity depends on the position control resolution and the IPO sampling time. The IPO sampling time is defined by MD 155. Standard value is 20 ms.

IPO sampling time [ms]	Position control resolution				
	$2^{-1}\ 10^{-2}$ mm	$2^{-1}\ 10^{-3}$ mm	$2^{1}\ 10^{-4}$ mm	10^{-4} mm	$2^{-1}\ 10^{-4}$ mm
	V_{max} [m/min]	V_{max} [m/min]	V_{max} [m/min]	V_{max} [m/min]	V_{max} [m/min]
16	614	61	24	12.2	6.1
18	546	54	21	10.8	5.4
20	491	49	19	9.7	4.9
22	446	44	17	8.7	4.4
24	409	40	16	8.0	4.0
26	377	37	15	7.4	3.7

IPO sampling time [ms]	Position control resolution				
	2^{-1} 10^{-3} inch	2^{-1} 10^{-4} inch	2^{1} 10^{-5} inch	10^{-5} inch	2^{-1} 10^{-5} inch
	V_{max} [inch/min]	V_{max} [inch/min]	V_{max} [inch/min]	V_{max} [inch/min]	V_{max} [inch/min]
16	61400	6140	2450	1220	614
18	54500	5450	2180	1090	545
20	49100	4910	1960	980	491
22	44600	4460	1780	890	446
24	40900	4090	1630	810	409
26	37700	3770	1510	750	377

Maximum velocities for linear axes

10.7.2.2 Coding of the resolutions with linear axes

- Input resolution

METRIC
MD 5002.4 = 0

Input resolution	MD 5002		
	Bit 7	Bit 6	Bit 5
10^{-2} mm	0	0	1
10^{-3} mm	0	1	0
10^{-4} mm	0	1	1
10^{-5} mm	1	0	0

INCH
MD 5002.4 = 1

Input resolution	MD 5002		
	Bit 7	Bit 6	Bit 5
10^{-3} inch	0	1	0
10^{-4} inch	0	1	1
10^{-5} inch	1	0	0
10^{-6} inch	1	0	1

= *Default*

- Display resolution

  The dimensional system is defined by the input resolution.

METRIC

Display resolution	MD 584*		
	Bit 7	Bit 6	Bit 5
10^{-2} mm	0	0	1
10^{-3} mm	0	1	0
10^{-4} mm	0	1	1

INCH

Display resolution	MD 584*		
	Bit 7	Bit 6	Bit 5
10^{-3} inch	0	1	0
10^{-4} inch	0	1	1
10^{-5} inch	1	0	0

- Position control resolution

METRIC
MD 5002.0 = 0

Position control resolution	MD 584*		
	Bit 3	Bit 2	Bit 1
$0.5 \cdot 10^{-2}$ mm	0	0	1
$0.5 \cdot 10^{-3}$ mm	0	1	0
$2 \cdot 10^{-4}$ mm	1	1	0
$1 \cdot 10^{-4}$ mm	1	1	1
$0.5 \cdot 10^{-4}$ mm	0	1	1

INCH
MD 5002.0 = 1

Position control resolution	MD 584*		
	Bit 3	Bit 2	Bit 1
$0.5 \cdot 10^{-3}$ inch	0	1	0
$0.5 \cdot 10^{-4}$ inch	0	1	1
$2 \cdot 10^{-5}$ inch	1	1	0
$1 \cdot 10^{-5}$ inch	1	1	1
$0.5 \cdot 10^{-5}$ inch	1	0	0

░ = *Default*

10.7.2.3 Combinations of resolutions with linear axes

- Permissible combinations of position control resolution and input resolution with linear axes

Position control resolution		Input resolution							
		METRIC MD 5002.4 = 0				INCCH MD 5002.4 = 1			
		10^{-2} mm	10^{-3} mm	10^{-4} mm	10^{-5} mm	10^{-3} inch	10^{-4} inch	10^{-5} inch	10^{-6} inch
MD 5002.0 = 0									
METRIC	$0.5 \cdot 10^{-2}$ mm	X	X	X	–	X	X	X	–
	$0.5 \cdot 10^{-3}$ mm	X	X	X	–	X	X	X	–
	$2 \cdot 10^{-4}$ mm	X	X	X	X	–	–	X	X
	$1 \cdot 10^{-4}$ mm	X	X	X	X	–	–	X	X
	$0.5 \cdot 10^{-4}$ mm	X	X	X	X	–	–	X	X
MD 5002.0 = 1									
INCH	$0.5 \cdot 10^{-3}$ inch	X	X	–	–	X	X	X	–
	$0.5 \cdot 10^{-4}$ inch	X	X	–	–	X	X	X	–
	$2 \cdot 10^{-5}$ inch	X	X	X	X	X	X	X	X
	$1 \cdot 10^{-5}$ inch	X	X	X	X	X	X	X	X
	$0.5 \cdot 10^{-5}$ inch	X	X	X	X	X	X	X	X

X Combination is allowed
– Combination is not allowed

- Combination of axis-specific display resolution depending on the position control resolution with linear axes

Position control resolution		Display resolution					
		METRIC			INCH		
		10^{-2} mm	10^{-3} mm	10^{-4} mm	10^{-3} inch	10^{-4} inch	10^{-5} inch
METRIC	$0.5 \cdot 10^{-2}$ mm	X	–	–	X	–	–
	$0.5 \cdot 10^{-3}$ mm	–	X	–	–	X	–
	$2 \cdot 10^{-4}$ mm	–	X	X	–	–	X
	$1 \cdot 10^{-4}$ mm	–	–	X	–	–	X
	$0.5 \cdot 10^{-4}$ mm	–	–	X	–	–	X
INCH	$0.5 \cdot 10^{-3}$ inch	X	–	–	X	–	–
	$0.5 \cdot 10^{-4}$ inch	X	–	–	–	X	–
	$2 \cdot 10^{-5}$ inch	–	X	–	–	X	X
	$1 \cdot 10^{-5}$ inch	–	X	–	–	–	X
	$0.5 \cdot 10^{-5}$ inch	–	X	–	–	–	X

X Combination is allowed
– Combination is not allowed

10.7.3 Combinations with rotary axes

10.7.3.1 Maximum velocities

The maximum rotary axis velocity depends on the position control resolution and the IPO sampling time.

IPO sampling time [ms]	Position control resolution				
	$2^{-1}\ 10^{-1}$ deg	$2^{-1}\ 10^{-2}$ deg	$2^{-1}\ 10^{-3}$ deg	$2^{-1}\ 10^{-4}$ deg[1)]	$2^{-1}\ 10^{-5}$ deg[1)]
	V_{max} [deg/min]	V_{max} [deg/min]	V_{max} [deg/min]	V_{max} [deg/min]	V_{max} [deg/min]
16	6140000	614000	61400	6140	614
18	5450000	545000	54500	5450	545
20	4910000	491000	49100	4910	491
22	4460000	446000	44600	4460	446
24	4090000	409000	40900	4090	409
26	3770000	377000	37700	3770	377

IPO sampling time [ms]	Position control resolution				
	$2^{-1}\ 10^{-1}$ deg	$2^{-1}\ 10^{-2}$ deg	$2^{-1}\ 10^{-3}$ deg	$2^{-1}\ 10^{-4}$ deg[1)]	$2^{-1}\ 10^{-5}$ deg[1)]
	V_{max} [rev/min]	V_{max} [rev/min]	V_{max} [rev/min]	V_{max} [rev/min]	V_{max} [rev/min]
16	17055	1705	170	17	1,7
18	15138	1513	151	15	1,5
20	13638	1363	136	13	1,3
22	12388	1238	123	12	1,2
24	11361	1136	113	11	1,1
26	10472	1047	104	10	1,0

Maximum velocities for rotary axes

1) NC MD 5031 bit 0 must be set for this

- Input resolution

METRIC
MD 5002.4 = 0

Input resolution	MD 5002		
	Bit 7	Bit 6	Bit 5
10^{-2} deg	0	0	1
10^{-3} deg	0	1	0
10^{-4} deg	0	1	1
10^{-5} deg	1	0	0

INCH
MD 5002.4 = 1

Input resolution	MD 5002		
	Bit 7	Bit 6	Bit 5
10^{-3} deg	0	1	0
10^{-4} deg	0	1	1
10^{-5} deg	1	0	0
10^{-6} deg	1	0	1

- Display resolution

The dimensional system is defined by the input resolution.

METRIC

Display resolution	MD 584*		
	Bit 7	Bit 6	Bit 5
10^{-1} deg	0	0	0
10^{-2} deg	0	0	1
10^{-3} deg	0	1	0
10^{-4} deg[1)]	0	1	1
10^{-5} deg[1)]	1	0	0

INCH

Display resolution	MD 584*		
	Bit 7	Bit 6	Bit 5
10^{-1} deg	0	0	0
10^{-2} deg	0	0	1
10^{-3} deg	0	1	0
10^{-4} deg[1)]	0	1	1
10^{-5} deg[1)]	1	0	0

1) NC MD 5031 bit 0 must be set for this

- Position control resolution

METRIC
MD 5002.0 = 0

Position control resolution	MD 584*		
	Bit 3	Bit 2	Bit 1
$0.5 \cdot 10^{-1}$ deg	0	0	0
$0.5 \cdot 10^{-2}$ deg	0	0	1
$0.5 \cdot 10^{-3}$ deg	0	1	0
$0.5 \cdot 10^{-4}$ deg[1)]	0	1	1
$0.5 \cdot 10^{-5}$ deg[1)]	1	0	0

INCH
MD 5002.0 = 1

Position control resolution	MD 584*		
	Bit 3	Bit 2	Bit 1
$0.5 \cdot 10^{-1}$ deg	0	0	0
$0.5 \cdot 10^{-2}$ deg	0	0	1
$0.5 \cdot 10^{-3}$ deg	0	1	0
$0.5 \cdot 10^{-4}$ deg[1)]	0	1	1
$0.5 \cdot 10^{-5}$ deg[1)]	1	0	0

1) NC MD 5031 bit 0 must be set for this

10.7.3.3 Combinations of resolutions with rotary axes

- Permissible combinations of position control resolution and input resolution with rotary axes

Position control resolution		Input resolution: METRIC MD 5002.4 = 0				INCH MD 5002.4 = 1			
		10^{-2} deg	10^{-3} deg	10^{-4} deg	10^{-5} deg	10^{-3} deg	10^{-4} deg	10^{-5} deg	10^{-6} deg
MD 5002.0 = 0									
METRIC	$0.5 \cdot 10^{-1}$ deg	X	X	–	–	–	–	–	–
	$0.5 \cdot 10^{-2}$ deg	X	X	X	–	–	–	–	–
	$0.5 \cdot 10^{-3}$ deg	X	X	X	–	–	–	–	–
	$0.5 \cdot 10^{-4}$ deg	X	X	X	X	–	–	–	–
	$0.5 \cdot 10^{-5}$ deg	X	X	X	X	–	–	–	–
MD 5002.0 = 1									
INCH	$0.5 \cdot 10^{-1}$ deg	–	–	–	–	X	–	–	–
	$0.5 \cdot 10^{-2}$ deg	–	–	–	–	X	X	–	–
	$0.5 \cdot 10^{-3}$ deg	–	–	–	–	X	X	–	–
	$0.5 \cdot 10^{-4}$ deg	–	–	–	–	X	X	X	–
	$0.5 \cdot 10^{-5}$ deg	–	–	–	–	X	X	X	X

X Combination is allowed
– Combination is not allowed

- Permissible combinations of display resolution and position control resolution with rotary axes

Position control resolution		Display resolution									
		METRIC (MD 5002.4 = 0)					INCH (MD 5002.4 = 1)				
		10^{-1} deg	10^{-2} deg	10^{-3} deg	10^{-4} deg	10^{-5} deg	10^{-1} deg	10^{-2} deg	10^{-3} deg	10^{-4} deg	10^{-5} deg
METRIC	$0.5 \cdot 10^{-1}$ deg	X	–	–	–	–	–	–	–	–	–
	$0.5 \cdot 10^{-2}$ deg	–	X	–	–	–	–	–	–	–	–
	$0.5 \cdot 10^{-3}$ deg	–	–	X	–	–	–	–	–	–	–
	$0.5 \cdot 10^{-4}$ deg	–	–	–	X	–	–	–	–	–	–
	$0.5 \cdot 10^{-5}$ deg	–	–	–	–	X	–	–	–	–	–
INCH	$0.5 \cdot 10^{-1}$ deg	–	–	–	–	–	X	–	–	–	–
	$0.5 \cdot 10^{-2}$ deg	–	–	–	–	–	–	X	–	–	–
	$0.5 \cdot 10^{-3}$ deg	–	–	–	–	–	–	–	X	–	–
	$0.5 \cdot 10^{-4}$ deg	–	–	–	–	–	–	–	–	X	–
	$0.5 \cdot 10^{-5}$ deg	–	–	–	–	–	–	–	–	–	X

X Combination is allowed
– Combination is not allowed

Notes for the incremental mode. The following table applies only if the input resolution has no larger grid. The table applies for linear axes and for rotary axes. The table states the pulse width for the increment, depending on the display resolution.

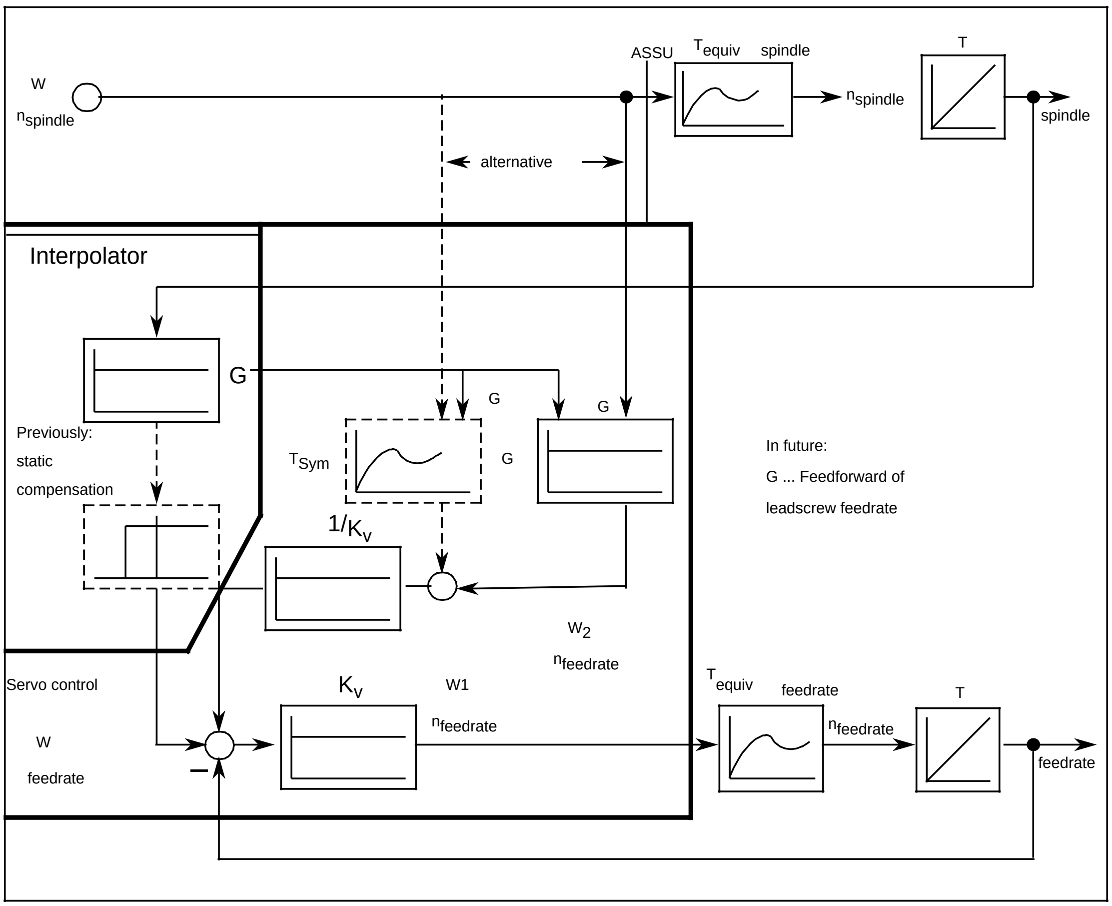
Display resolution / Increment	10^{-1} deg	10^{-2} deg (mm)	10^{-3} deg (mm) (inch)	10^{-4} deg (mm) (inch)	10^{-5} deg (mm) (inch)
INC 1	0.1	0.01	0.001	0.0001	0.00001
INC 10	1	0.1	0.01	0.001	0.0001
INC 100	10	1	0.1	0.01	0.001
INC 1000	100	10	1	0.1	0.01
INC 10000	1000	100	10	1	0.1

10.8 Tapping with dynamic following error compensation (optional)

10.8.1 Brief description

- In tapping, because the feed axis is coupled to the main spindle actual value, the feedrate setpoint can only change with a delay of 1.7 IPO cycles. This results in a position deviation at the drill which can lead to tool breakage under certain operating conditions when the speed changes during reversal.
- The main innovation was, therefore, to make the feedrate setpoint dependent on the HSP setpoint, see block diagram.

  With the new function the feedrate setpoint (n^*_{HSP} ·pitch) is output synchronously, and therefore error-free, with the HSP setpoint.
- The spindle actual value branch has been retained. It is required to regulate actual value deviations on the controlled system (statically!). However, the problem mentioned above still applies when reversing. This has been improved by introducing a dynamic dead-time compensation and halving the influence active at the feedrate setpoint. The main disadvantage, however, that all controlled-system influences only act with a 38 ms delay still applies.



Block diagram of tapping with dynamic compensation

10.8.2 Description of the new machine data

MD 256* **Difference time constant**

Unit:	0.1 ms		
Standard:	0.2 ms		
Input limits:	0.0 ms		No adaptation
	0.1 ms	to 2.7 ms	Time-lag element
	2.8 ms	to 8.2 ms	p element
	8.3 ms	(to 999.9) ms	pT2 element

With this machine data the dynamic behaviour (inertia) of the axis is adapted to the spindle (pT2 element).

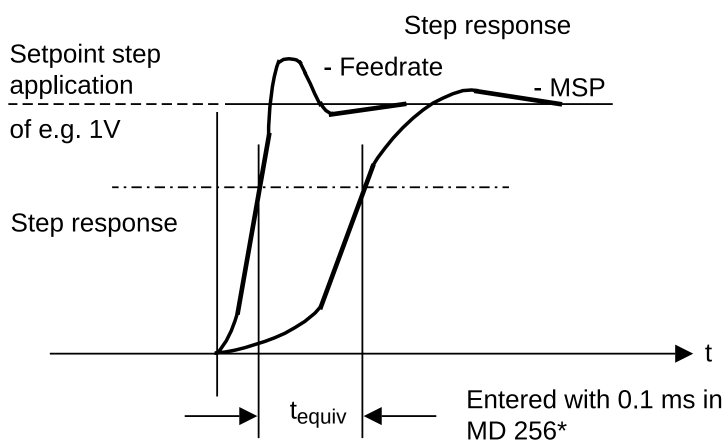
Note:

First determine the time constants of the feed axis and the main spindle and then enter the difference time constant T_{equiv} into machine data 256*.

Acquisition of the delay (T_{sym}) for the drilling axis

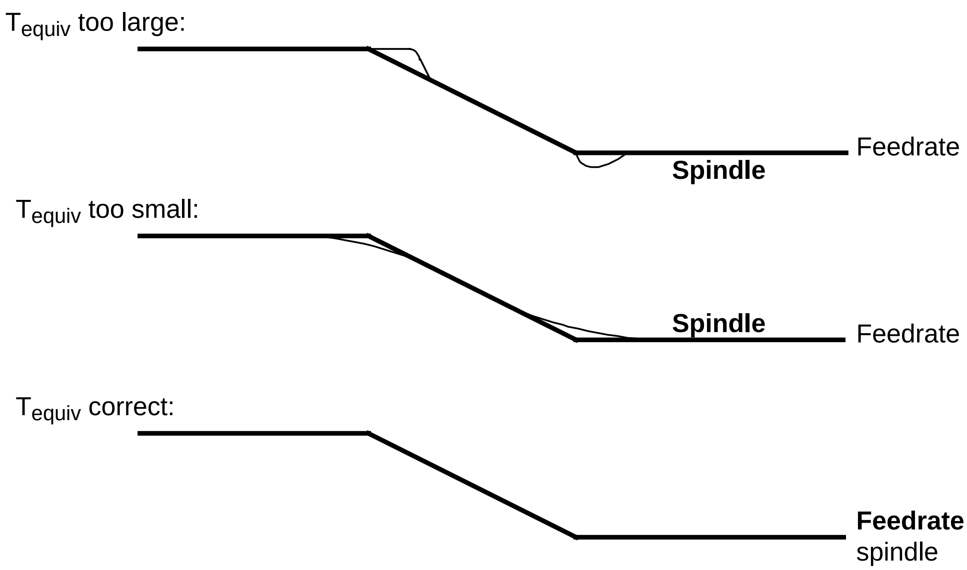
Here the dynamic response of the speed control-loop of the drilling axis is adapted to the HSP so that the actual value of the spindle and the axis agree.

T_{equiv} can clearly be seen from the evaluation of the step response of the spindle and drilling axis.



This measurement must be performed when starting up the prototype. The value can be used for series production.

Checking the speed actual value variations

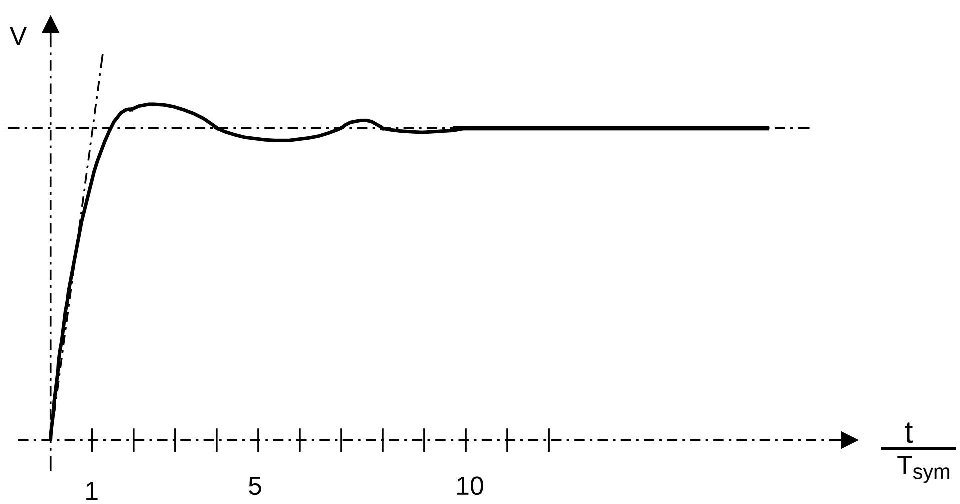


MD 360* Symmetrizing time constant (T_{sym})

Unit: 0.1 ms (Standard: 0.0 ms)

Input limit:	0.0 ms	to	2.7 ms	Time-lag element
	2.8 ms	to	8.2 ms	P element
	8.3 ms	to	999.9 ms	PT2 element

With this machine data, issue of the feedrate setpoint on starting and braking is delayed (PT2 element, acceleration is completed within time $10 x T_{sym}$).



In order to be sure that the axis has accelerated before contact with the workpiece when tapping, a safety distance (s) must be taken into account in the program:

$\boxed{s = 10 \cdot T_{sym} \cdot l \cdot n}$ or as an adapted equation:

$$\boxed{s_{[mm]} = \frac{T_{sym\,[0.1\,ms]} \cdot l_{[mm]} \cdot n_{[min^{-1}]}}{6000}}$$

T_{sym} is input in 0.1 ms. This obviates multiplication (*10) in the NC software (runtime optimization)

s: Thread run-in path (safety distance) [mm]
T_{sym}: Symmetrizing time constant [0.1 ms]
l: Thread pitch [mm]
n: Spindle speed (with which tapping is performed) [rev/min]

Example of calculation of the run-in path for M2 thread:

T_{sym} = 10 ms
l = 0.4 mm
n = 3000 rev/min

$$s = \frac{100 \cdot 0.4 \cdot 3000}{60\,000} = 2$$

s = 2 mm

Example of calculation of the run-in path for M20 thread:

T_{sym} = 10 ms
l = 2.5 mm
n = 500 rev/min

$$s = \frac{100 \cdot 2.5 \cdot 500}{60\,000} = 2.08$$

s = 2.08 mm

As can be seen from the two examples, the safety distance is practically the same regardless of whether M2 or M20 is being drilled.

The safety distance determined on start up must be entered in the programming guide for the machine.

The ”knock” on starting and stopping the axis is audibly reduced as from approx. 10 ms (MD 360*) .

10.8.3 Note the following NC data

- **MD 220* Precise setting of backlash compensation**

- **MD 260* Multgain**

  For dynamic compensation the multgain factor **must** contain the exact compensation value with the drive.

- **MD 252* Servo gain factor (K_V factor)**

  On warm restart the servo gain factor is calculated for the compensation logic. The restart routine loads the first servo gain factor with a value other than zero.

  Thread axis must be defined as the first axis if different servo gain factors are used for the axes.

- **MD 520* bit 1**
  MD 521* bit 1

  The spindle bits sign inversion setpoint and sign inversion actual value have to be set so that a positive actual speed value appears on M3 programming in the display.

- **Spindle setting data for smoothing constant**

  For threads it must be zero.

- **Use tapping cycle L84 (contained in UMS42).**

  This cycle has overrun path compensation.
  For a blind hole thread, the programmed thread path must be reduced because the feed axis is rigidly coupled to the main spindle. An overrun path arises on reversal which is calculated in cycle L84 and is subtracted from the programmed path. The cycle also tests the final path and provides a minimum traverse path (overrun path). If the path originally programmed is shorter, the cycle reduces the spindle speed until the overrun path is equal to the original thread path.

10.9 Block transfer (options C69)

The ”Block transfer” function is used for complex machining when the program can have a length of 20 to 30 Mbytes in, for example, 3D milling. Because the memory of SINUMERIK 810 GA3/820 GA3 can only be expanded to 128 Kbytes, a part program of this size is executed using this function (on channel 1 only).

10.9.1 Mode of operation

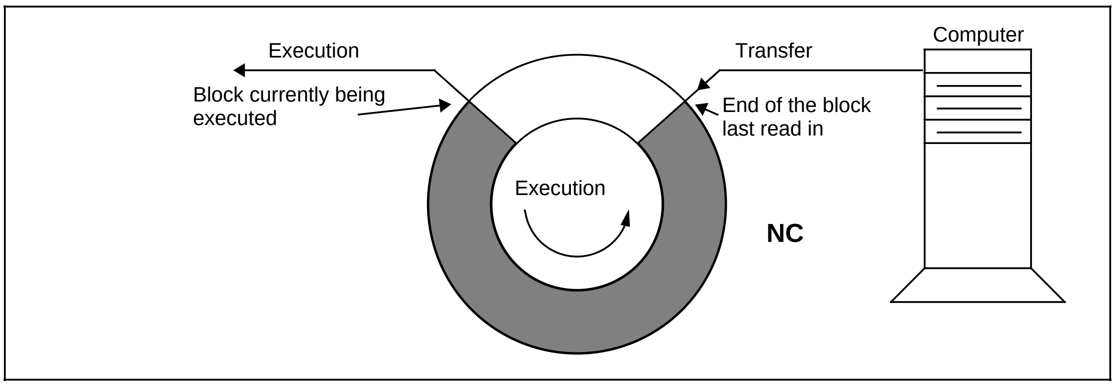
Memory area: The memory area used is the so-called ”circular buffer” which can be set variably in 1 Kbyte steps via NC MD 20, the upper limit being 64 Kbytes.

– A change of NC MD 20 becomes active only after ”Clear part program memory”.
– Note that the circular buffer reduces the memory area for part programs.

– Calculation blocks between traversing blocks must not exceed or reach the length of the circular buffer because the NC does not reset flag 23.7 (circular buffer full) in calculation blocks.
– The circular buffer cannot be displayed on the NC monitor or edited. However, the number of blocks in the circular buffer is entered in FW 19 or FW 21 of the PLC in hexadecimal format.

FW 19 $\hat{=}$ number of blocks (KF) in the first program
FW 21 $\hat{=}$ number of blocks (KF) in the second program

Circular buffer management: The program blocks transferred are loaded into the circular buffer and executed from there. While the program is running, areas of the buffer become free and are automatically loaded with new blocks.



Schematic of data transfer using the circular buffer

Flag 23.7 is set when the circular buffer is full.
Flag 23.6 is set when the circular buffer is empty.
These flags are automatically reset by the control.

Selection: Block transfer is selected in the menu with softkey BTR START. The BTR function can also be selected via the PLC in DB 37. This is done via bit 7 (computer link) of DL 1. The expected program number is entered in ANFNR DW4. If zero is entered in DW4 every main program is read in as with the softkey function without any checks being made as to whether the read-in main program number corresponds to the main program number entered in the automatic mode display.

Program no.:
– The program number is entered into the basic display of automatic mode.
– Only main programs are permissible.
– If the selected program number is unequal to the program number read in, Alarm 33 appears.
– EXCEPTION: When program number ”%0” is input, the program number which arrives is not checked.
  This means that a program can, for example, be executed from paper tape without the user having to know the program number. However this program must also be identified as a main program (%MPF...).

Program modification:	All functions such as decoding, single block, read-in enable, etc. are possible just as in normal operation.
Block search:	Block search is possible without restrictions. Normal operation is resumed with NC START.
Sequence of operation:	1. Program number entered in basic display of automatic mode (channel 1) 2. Enter interface assignment (do not start interface). 3. Activate ”Block transfer” 4. Press NC START

10.9.2 Sources of error for the ”Block transfer” function

- In the main program, @ commands with a return jump are not allowed (Alarm 3012).
- With forward jumps it is important to take into account that the corresponding block may not yet be in the circular buffer and must first be loaded.
- Subroutine calls are only possible if the subroutines are either in the NC part program memory or in the UMS. If this is not the case, Alarm 2041 appears.
- If a program other than the selected program is transferred by the sender (exception %0) this causes Alarm 33.
- If ”Block transfer” is selected when an interface is already active, the message ”Interface busy” appears. The program is not started.
- If ”Block transfer” is active and the other interface is started by the PLC (DB 37), the message ”V.24 ABORT” appears.The program continues.
- It is not possible to edit the circular buffer. If a programming error is discovered, pressing the ”EDIT BLOCK” softkey will lead to the message ”NO EDIT BLOCK”.

10.9.3 Procedures

Data transfer is either unprotected or protected.

10.9.3.1 Unsecured data transfer

Unsecured data transfer is performed with Xon/Xoff via the serial interface selected.

If the unassigned area in the circular buffer is less than120 bytes, the signal ”circular buffer full” (M23.7) is set.

If the circular buffer is empty, the signal ”circular buffer empty” (M23.6) is set.

Unsecured block transfer is intended for external memory units. It can be used for example, for connecting a DSG 3.5 diskette drive which is available as an accessory to the SINUMERIK 810/820 GA3.

Transfer is initiated by the NC sending Xon to the memory unit. The memory transfers data until it is stopped by the NC with the Xoff signal (circular buffer full). As soon as the circular buffer is empty, the external memory is activated with Xon to send the next data.

10.9.3.2 Secured data transfer

Secured reading-in can be selected via machine data MD 5016.1=1. After secured reading-in has been selected, data transfer is initiated with protocol 3964R. The precondition for this is that the sender understands this protocol and suitably packs the data to be sent.

Data transfer is started by the SINUMERIK 810/820 GA 3. After NC start, the control sends a command block to the external computer. The command block also contains the required program number so that data transfer need not be started at the external computer. The external computer starts sending data on receipt of the initialisation block.

10.9.3.3 Data transfer procedure 3964R

Procedure 3964R is an asynchronous, bit-serial transfer procedure. Sending and receiving clock frequency (baud rate) must be set to the same value since there is no clock line between the two devices (asynchronous operation).

Procedure 3964R handles all activities for controlling the data flow with another computer. A program which evaluates the data, transfers transmitted data to the procedure in an output buffer provided for that purpose. The procedure sends these data with the corresponding transmission protocol 3964R, repeats the transmission if necessary and reports to the calling program those errors that cannot be remedied. The received data are stored in an input buffer provided for that purpose and – if correctly received – are transferred to the data interpreter for further processing.

Control and useful information characters are sent via the communication line. Further bits are put in front of or behind the transmitted character to be able to recognize every character at the receiver and to check for error-free transmission. The sequence of the bits on the line is as follows:

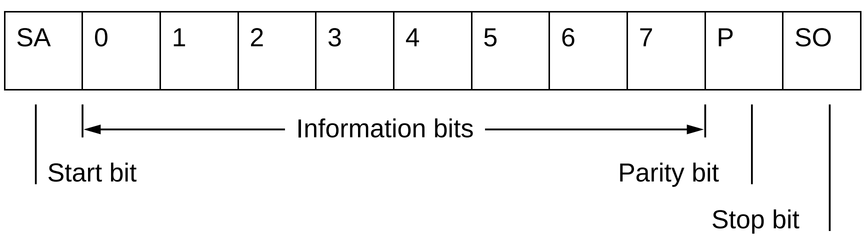


Figure 5: Procedure 3964R character sequence

The characters transmitted are stored with even parity. In addition, a block check character is transmitted at the end of the block according to the character parity procedure. Data acquisit-ion includes all characters of the block except the STX character.

The control characters for procedure 3964R are in accordance with DIN 66 003 for the 7-bit code. They are transmitted with a character length of 8 bits (bits 0 to 7, bit 7=0).

The following information characters are used:

STX	02H
DLE	10H
ETX	03H
NAK	15H

10.9.3.4 Transmitting with procedure 3964R

The procedure transmits the control character STX to establish a connection. Transmission can be started if the I/O device responds with the DLE character before 2 seconds have elapsed. A connection cannot be made if no response is received within the specified time or a different character is sent. After three vain attempt the procedure is aborted and sends an error message to the calling program.

If the connection is successful, the received characters are written into the input buffer provided for that purpose. The characters contained in the output buffer are prepared by the procedure 3964R and sent to the I/O device.

After transmission of the output buffer the procedure adds the characters DLU, ETX and BCC as end identifiers and waits for an acknowledgement character. If the I/O device does not respond within 2 seconds or with a character other than DLE, the procedure repeats transmission of the data block. After 3 vain attempts to send the data block, the procedure stops the process and signals an error. If the I/O device sends as DLE character within 2 seconds, this means that the data block has been transmitted without an error.

10.9.3.5 Receiving with procedure 3964R

If the procedure 3964R receives the STX character and has an empty input buffer available, it responds with DLE.

The subsequent characters are now written into the input buffer. The arriving characters must be sent at an interval of 220 ms or less. If the interval between two characters is greater than that, the transmission is considered to be faulty (NAK to transmitter on acknowledgement). The individual characters are checked for parity. In the event of incorrect parity, the transmission is reported to the sender as being faulty (NAK).

The procedure terminates reception when it recognizes the character sequence DLE ETX BCC and sends DLE for error-free or NAK for faulty reception. If the block cannot be received without an error after the third attempt or if repetition is not started by the I/O device within 10s, the procedure 3964R stops the process and signals an error.

10.9.3.6 Message frame

The message frame has the following format

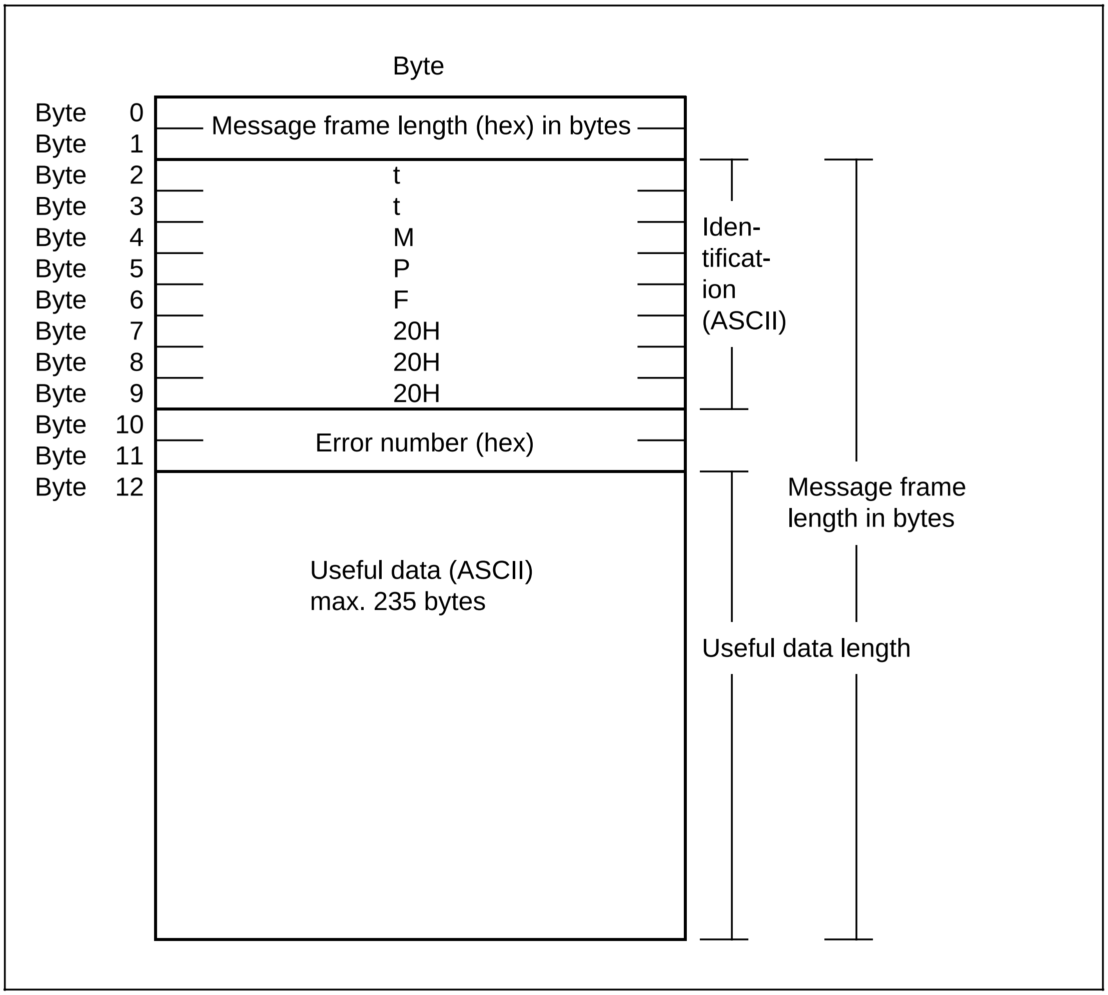


Figure 6: Message frame with header

Identification:

MPF: Main program
Request TN for main program
Receive RN for main program
tt: Message frame identifier

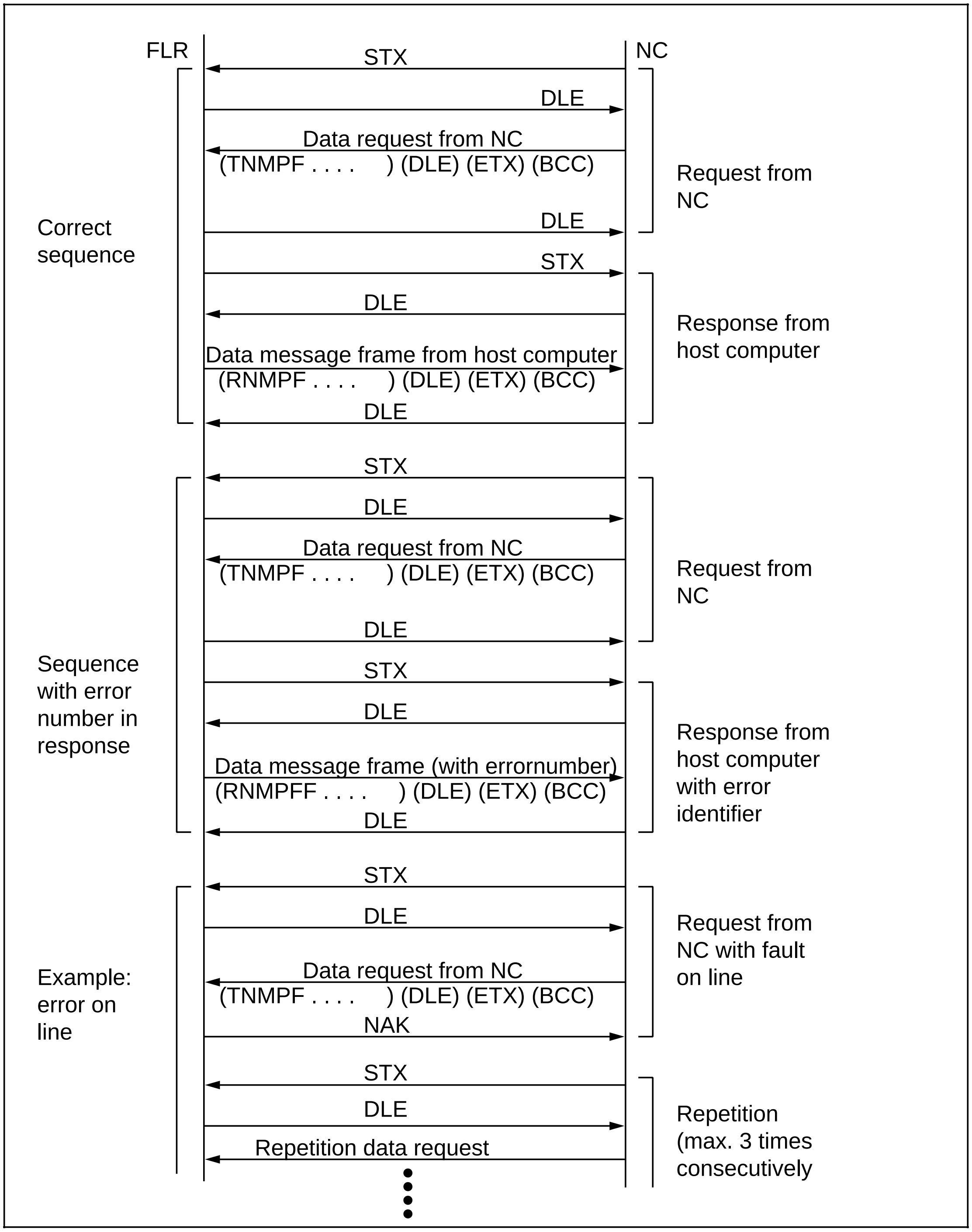
10.9.3.7 Message frame interchange between host computer and NC

Interactive mode between NC and host computer without control characters.

Function	Host computer NC
1. NC requests data for next message frame	TNMPF_ _ _fnr[<vvvv bbbb>] ←
2. Acknowledgem. by host comp. a) Negative acknowledgem. with error number and abort of interactive mode	TNMPFF_ _fnr[<vvvv bbbb>] →
b) Transm. of NC data blocks from host computer to NC	RNMPF_ _ _fnr[<vvvv bbbb>] <%MPF1 LF . . >] →
3. NC requests the host computer to send the next message frame (following message frame)	TNMPF_ _ _fnr[<vvvv bbbb>] ←
4. Repetition of 2. and 3. if further following message frames are transmitted	(Repetition loop) (↔)
5. End of NC data transmission (message frame may or may not contain useful data)	RNMPF _E_fnr[<vvvv bbbb>] < . . . M02 LF>] →

vvvv: Program number
bbbb: Frame number is incremented with every new request message frame
fnr: Two-digit, 00 if no error

Interactive mode between NC and host computer with control characters



10.9.4 Message frame error messages

The following list shows the errors indicated to the operator during file transfer from the host computer. The error numbers are divided into groups.

Error no.	Error groups
1 ... 19	General errors during message frame input
20 ... 39	Errors during NC file transfer – displayed as NC alarm (corresponding to RS232C (V.24) alarm)
200 ... 299	General errors during message frame output
3100 ... 3149	Errors during NC file transfer

Error list:

Error no.	Error groups	NC alarm
26	Part program block >120 characters	28
29	Block too long < max. 254 characters	29
32	Data format errors	32
33	Program stored Program entered	33
36	BTR aborted by computer	36

10.10 Integrated tool management (Option N05)

Precondition:

- Option N31 or N32 ”Configurability”.
- Option F72 ”External data input”

10.10.1 Mode of operation

The integrated tool management is intended for use on machine tools equipped with chain-type or rotary-plate magazine and supports tool change with choice of direction of rotation. This option allows extremely fast tool change, avoids rejects by monitoring tool applicability and prevents machine downtimes by allowing for the use of spare tools.

The tool data can be input either via the operator panel or via the serial interface.

The PLC function blocks required for integrated tool management are accommodated in a separate macro memory area and do not reduce the PLC program memory space. The relevant screen forms are integrated in the system.

10.10.2 Technical data

- Management of up to 97 tools
- Use of spare tools
- Workpiece count
- Tool life inspection
- Flexible location coding for oversize tools
  (also selectable for normal-size tools)
- Screen forms for
  - Tool data input
  - Display of tool data
  - Display of disabled tools

No further details on the function of tool management are given here because there is a special Planning Guide on this subject.

Order number: 6ZB5 410-0DX02-0A...

10.11 Host PLC link

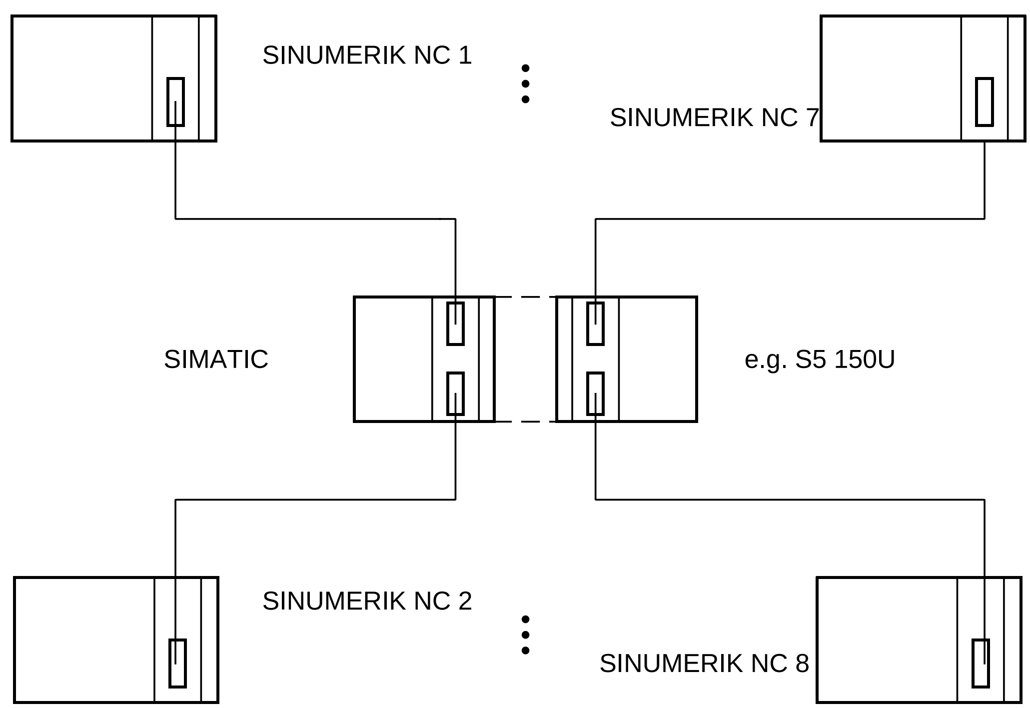
10.11.1 General notes

Up to 8 individual SINUMERIK controls can be linked via a host controller (SIMATIC S5). To link two SINUMERIK controls, one link module is required for each (two interfaces) in the SIMATIC (see Section 10.11.2). A transmission line of up to 150 m is possible.

Data exchange for the 8-bit link is achieved via a dualport RAM (DPR) on the link module in the SIMATIC. To send and receive data on the SINUMERIK side, the function macro FB 190 (K-LEITPC) must be parametrized and called accordingly, on the SIMATIC side the function block FB 190 is used.

10.11.2 System configuration

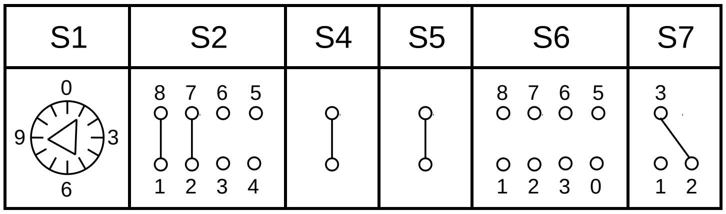
Up to 8 external units (SINUMERIK controls) can be connected to the host PLC (e.g. S5 150U).



Host PLC link module (in SINUMERIK), 6FX1135-6BA
Host PLC link module (in SIMATIC), 6FX1135-7BA
Cable for PLC-PLC link, 6FC9 340-7H

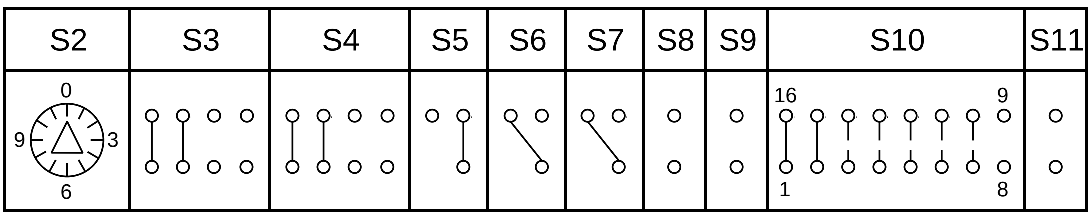
10.11.3 Jumpering on the link modules

The 6FX1135-6BA link module (in SINUMERIK) has the following standard job jumperings:



S1: With the rotary switch S1 (on the front panel), the size of the link area is set (on SINUMERIK position 1 ≙ 1/4 Kbyte).

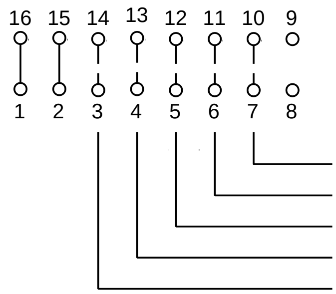
The 6FX1135-7BA link module (in SIMATIC) has the following jumpering options:



S2: With the rotary switch S2 (on the front panel), the address area I, II, III, or IV (0.5 Kbytes each) within the dual port RAM (2 Kbytes) is assigned to the corresponding module. The initial address of the dual port RAM (DPR) is set via S10 and depends on the type of control (see S10).

Example:
Six SINUMERIK controls are to be connected to the host PLC (e.g. S5 150U). For this system configuration three link modules are required in the S5 150U.

S10: The initial address of the dual port RAM (DPR) can be set in two Kbyte steps via S10 (jumpers 3 -14 to 7-10 shown with a broken line).

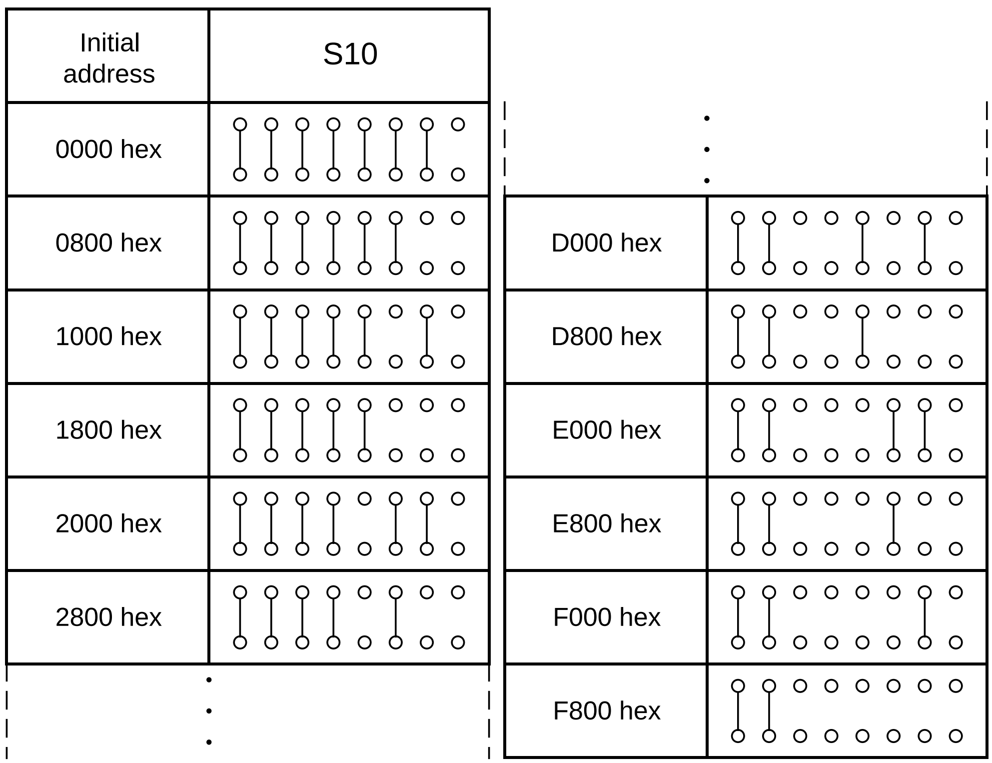


Significance of the jumpers

2^{11} = 2 048 (2 Kbytes)
2^{12} = 4 096 (4 Kbytes)
2^{13} = 8 192 (8 Kbytes)
2^{14} = 16 384 (16 Kbytes)
2^{15} = 32 768 (32 Kbytes)

Jumper **not** inserted significance is active

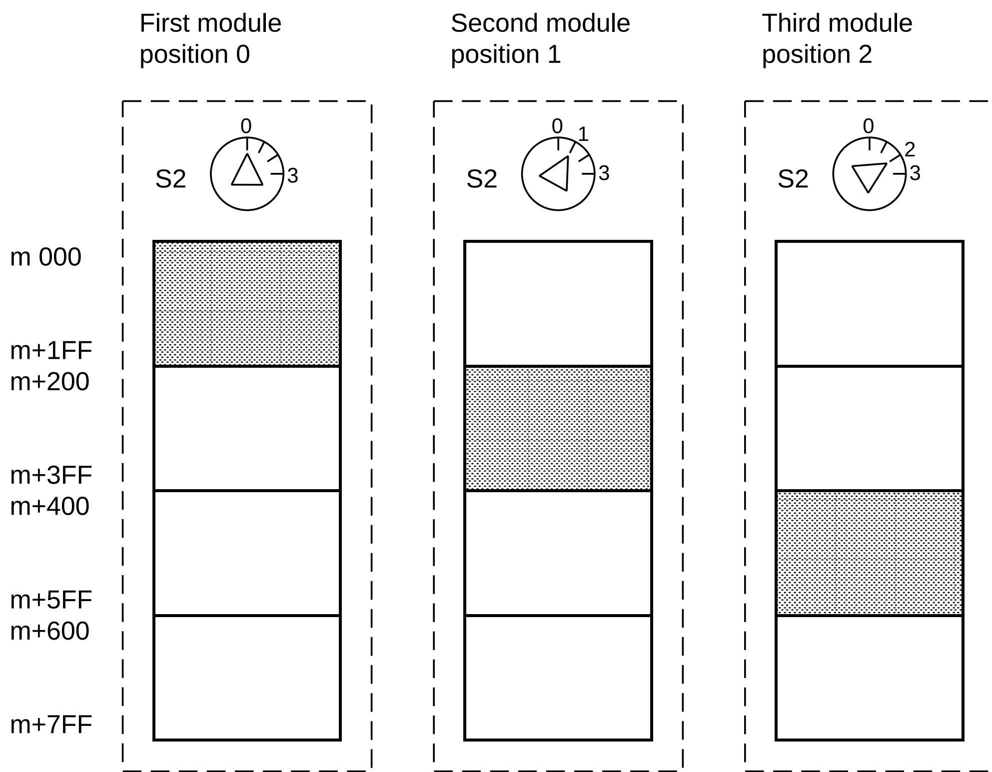
Example:



Addressing in the host PLC

The initial address of the DPR is set via S10 on all modules (e.g. on E000 hex initial address of the DP on S5 150U).

Switch 2 must be put into these positions on the following modules:



This defines address areas I, II,III and IV of the DPR as follows:

Address Area	Assignment
	1st module
	2nd module
	3rd module
	free for a further module or WF 470

For ”m” see ”Table of permissible address areas”.

The following table shows the permissible address areas for the dual-port RAM (DPR) for various PLC types.

PLC type	Address area for DPR
S5-115U	0000 hex to 0FFF hex (priority area)
S5-135U	F800 hex to FBFF hex The area from FC00 can only be used if no I/O module is plugged into the central controller
S5-150U	E000 hex to E7FF hex (priority area) F800 hex to FDFF hex
S5-155U	0000 hex to CFFF hex
S5-130W	E000 hex to E7FF hex (priority area)

The address areas F200 hex to F7FF hex can always be used if no CP modules are used.

10.11.4 Function blocks, error messages

The data is transferred by function block FB 190 which in the SINUMERIK is contained as a function macro in the operating system. FB 190 on the SIMATIC side is contained on the diskette 6MF1100-7AB00.

Parameters of FB 190

Parameter	Comment	Type	For-mat	Permissible values
ADPR	Initial address of the dual port RAM	D	KH	See address areas for control type
DBDW	DBNR/1 data word	D	KY	Data block no.2 up to 255
RI/A	RI = 0 ≙ send RI = 1 ≙ read A = Quantity of data words	D	KH	RI = 0 or 1 Quantity 1 to 126
MELD	A	A	KH	FY 0 to 199 QB 0 to 127

With the parameter ”MELD”, error numbers appear which must be evaluated by the user. If an error message appears, no data transfer takes place.

Error 1: Interface faulted
Error 2: Data block missing
Error 3: Data block is too short
Error 4: DB number not permissible(e.g. DB 0 or DB 1)
Error 5: DW 0 or DW 127 not permissible in the data block
Error 6: Parameters for RI (Richtung = direction) not permissible

If no error is present, the value FF_H is written in parameter ”MELD”. As long as FF_H is in the parameter ”MELD”, no check is made to see whether errors 2 to 6 are present. If this check, for example during installation, is to run continuously the flag byte specified in parameter ”MELD” must be set to 00_H before calling the FB 190.

It is advisable to set this set byte to 00_H after installation in the cold or warm restart branch. Then a check is made for errors 2 to 6 in the first pass.

The FB available in the SINUMERIK has the same parameters but the information given in parameter ADPR has no significance.

With the ADPR parameter of the SIMATIC S5 block, it is important to note that the initial address always specifies the beginning of the 1/4 Kbyte area. For example, if the module is addressed at $E100_H$, the first 1/4 Kbyte area starts at address $E200_H$, the second 1/4 Kbyte area for the second interface at $E300_H$.

10.11.5 Example

Words 10 to 30 are to be transferred from data block DB 33 in a SINUMERIK to the same data words in data block DB 45 in a SIMATIC. Likewise, data words 100 to 120 are to be transferred from the SIMATIC to the SINUMERIK.

The initial address of the dual port RAM is set at E000 hex via jumpers and the SINUMERIK is connected to the second interface (initial address = E000 hex + 100 hex = E100 hex).

PLC Programm

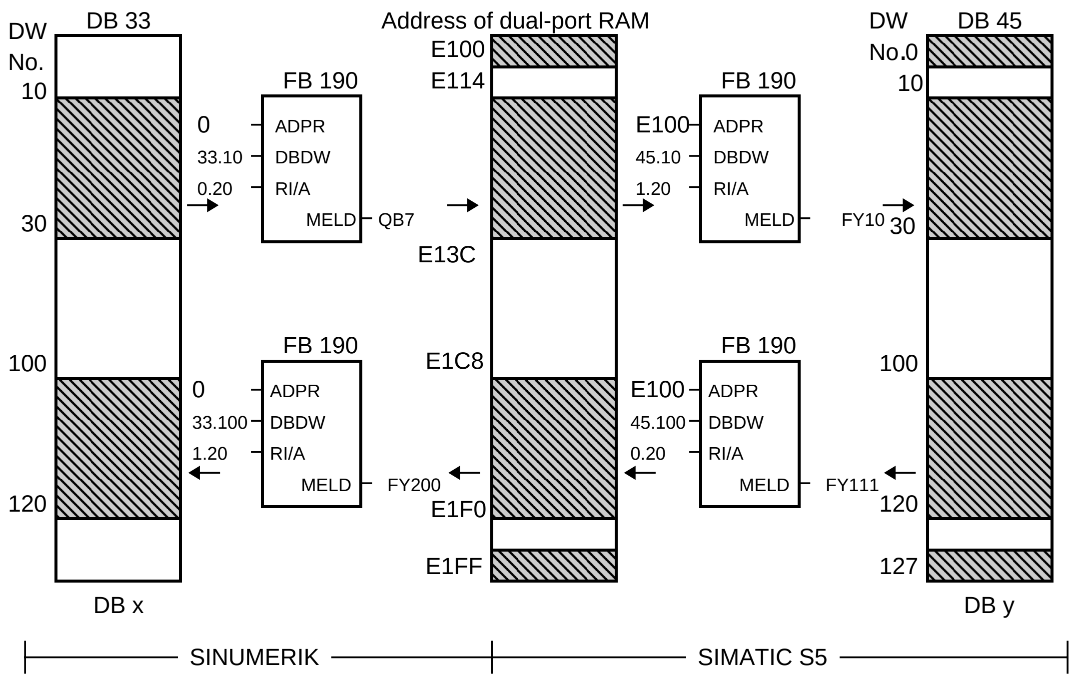
in the SINUMERIK

```
      :
      :  JU FB 190
NAME  :  K-LEITPC
ADPR  :    KH 0000
DBDW  :    KY 33,10
RI/A  :    KY 0,20
MELD  :    FY 180
      :
NAME  :  K-LEITPC
ADPR  :    KH 0000
DBDW  :    KY 33,100
RI/A  :    KY 1,20
MELD  :    FY 181
```

in the SIMATIC

```
      :
      :  JU FB 190
NAME  :  S5-NC810
ADPR  :    KH E100
DBDW  :    KY 45,10
RI/A  :    KY 1,20
MELD  :    FY 10
      :
NAME  :  S5-NC810
ADPR  :    KH E100
DBDW  :    KY 45,100
RI/A  :    KY 0,20
MELD  :    FY 11
```

As shown in the example, FB 190 must be called twice in every control. A call is necessary to ”Send” and call is necessary to ”Receive”. In the SIMATIC this double call of FB 190 is necessary for each interface.



As shown in the diagram, one standard function block FB 190 transfers the data block marked I from DB 33 in the SINUMERIK to DB 45 in the SIMATIC and the block marked II in the reverse direction.
The required block length is defined by parameter RI/A.

The position of the data block in the link area is determined by the initial words of the data block. DW 0 and DW 127 must not be used. Between these two data words the data block can be freely positioned.

It is up to the user to set the parameters of the data blocks. He can write data block I of DB 33 with load and transfer commands in a program and, in this way, define which functions and messages are to be transferred to the SIMATIC S5 and which are to be processed there.

Feedback messages from the SIMATIC S5 are transferred via data block II.

The internal PLC of the SINUMERIK must not be replaced by the linked SIMATIC S5. In transfer lines, the SIMATIC S5 has the function of a host controller. For example, after switching on the plant, all SINUMERIK controls are switched to ”Reference point approach” mode from the central control panel and then the automatic program of the SINUMERIK controls can be preselected and started.

The diagram shows the data flow between the FB 190 function macro on the SINUMERIK side and the function block on the SIMATIC side via the link area.

The number of data words to be transferred both for the FB 190 function macro on the SINUMERIK side and for the FB 190 function block on the SIMATIC side must be the same for the two communicating function blocks FB 190. The number of the first data word on the SINUMERIK side must be identical to the number of the first data word on the SIMATIC side.

10.12 Coordinate transformation

The coordinate transformation TRANSMIT is used in the face milling of turned parts (turning machine). In order to implement this, a C axis and a driven milling cutter are required in addition to the X and Y axes.

10.12.1 Corresponding data

- NC MD 5061 G function for transformation selection
  NC MD 5062 Axis name 1st fictitious axis
  NC MD 5063 Axis name 2nd fictitious axis
  NC MD 5064 Axis name of infeed axis
  NC MD 5065 Axis name 1st real axis
  NC MD 5066 Axis name 2nd real axis
- MD 540* Bit 7 no cancelation of transformation with RESET
- MD 564* Bit 6 fictitious axis
- OPTION transformation TRANSMIT
- Signal E103.7 TRANSFORMATION ACTIVE
- Alarm 2043 Program error with transformation
- Alarm 2056 Traverse through transformation centre
- Alarm 2189 Transformation undefined
- Alarm 2190 Transformation axes assigned
- Alarm 2191 Transformation in zero
- Alarm 3083 Feedrate limitation fictitious axis
- Alarm 3087 Error in transformation data

10.12.2 Function description of coordinate transformation

Whereas machine movements are executed in the real machine coordinate system, programming is carried out in the fictitious (Cartesian) coordinate system. Fictitious axes must be defined especially for the fictitious coordinate system. A fictitious axis can only be traversed when transformation is selected. Fictitious axes can be selected freely with respect to their axis name and their location.

Transmit is selected in the part program using G131.

The transformation is permanently assigned to channel 1.

Coordinate transformation is cancelled by G130 in the part program.

It can be specified with NC MD 540* bit 7 whether cancelation is automatically performed by the NC on key RESET or after a change in the operating mode.

See Section 10.12.7 for applications of the coordinate transformation TRANSMIT.

If the options ”Transmit” and ”Cam signals” are active in the control at the same time, only real axes can be assigned to the software cams.

10.12.3 Transformation data record

NC MD	Bit no. 7	6	5	4	3	2	1	0
5061		G function for transformation selection						G131
5062		Axis name 1st fictitious axis						
5063		Axis name 2nd fictitious axis						
5064		Axis name infeed axis						
5065		Axis name 1st real axis						
5066		Axis name 2nd real axis						

Conditions for a transformation data record

a) The transformation option must be available.
b) The transformation data are taken over internally by the control on warm restart. Definition errors cause alarm 3087 to be output. The incorrect machine data is coded in the block number in the alarm text.
c) NC MD 564* bit 6 must be set for all fictitious axes.
d) The axis names of real and fictitious axes of a transformation data record must not overlap. Using the same axis name with a different extended address is permitted.

7	6	5	4	3	2	1	0	Bit No.

Extended address (address number)	Address name
Key:	Key:
blank = 0000	X = 0000
1 = 0001	Y = 0001
2 = 0010	Z = 0010
3 = 0011	A = 0011
4 = 0100	B = 0100
5 = 0101	C = 0101
6 = 0110	U = 0110
7 = 0111	V = 0111
	W = 1000
	Q = 1001
	E = 1010

Example: 0000 0010=Z
0001 1001=Q1

Legal names for axes, angles, chamfer and radius

A	*unassigned address*	**N**	block start
B	*unassigned address*	**O**	danger of confusion with 0 (zero)
C	*unassigned address*	**P**	subroutine-number of passes
D	tool offset number	**Q**	*unassigned address*
E	*unassigned address*	**R**	calculation parameter
F	feedrate	**S**	spindle speed, S function
G	G function	**T**	tool
H	H function	**U**	*unassigned address*
I	interpolation parameter	**V**	*unassigned address*
J	interpolation parameter	**W**	*unassigned address*
K	interpolation parameter	**X**	*unassigned address*
L	subroutine	**Y**	*unassigned address*
M	M function	**Z**	*unassigned address*

10.12.4 Machine data for fictitious axes

MD 224*		Software limit switch
MD 228*		Software limit switch
MD 232*		Software limit switch
MD 236*		Software limit switch The software limit switch need not be input if the fictitious working area is outside the real possible working area, as the control always restricts the fictitious software limit switch to the limit switch of the A_{1R} axis (linear axis of transformation).
MD 276*		Acceleration The acceleration value must be calculated in such a way that the real axes of transformation are not overloaded (acceleration value of A_{1R}).
MD 280*		Maximum speed
MD 288*		JOG speed
MD 292*		Rapid JOG The speeds can be freely selected, as they are monitored by the control.
MD 300*		Incremental speed
MD 304*		IPO parameter
MD 564*	Bit 6	Fictitious axes The axis is declared as a ”fictitious axis”. Fictitious axes have no position control. The MD 200* measuring circuit assignment must therefore be zero.
MD 564*	Bit 7	Axis exists
MD 568*		Encoding of the axis name

10.12.5 NC PLC interface signals

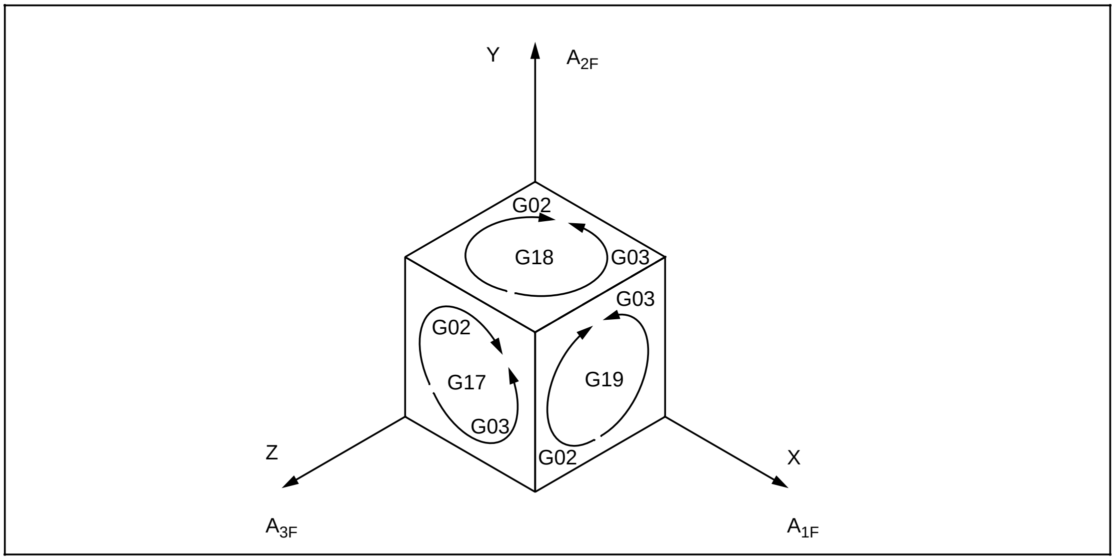
- The PLC signals ”Mirroring”, ”Feedrate enable”, ”2nd software limit switch active”, ”Rapid override”, ”Travel commands” and ”Handwheel 123” act on fictitious axes in the same way as real axes that are not part of the transformation grouping.
- When transformation is selected, the PLC outputs ”Mirroring”, ”Rapid override”, ”Travel commands”, ”Handwheel 123” and ”Delay” are not evaluated for the real axes involved.
- The PLC outputs ”Delay”, ”Parking axis” and ”Axis disable” have no influence on fictitious axes.
- The inputs ”Axis under position control” and ”Exact stop coarse/fine” are cancelled if a fictitious axis is switched to follow-up mode. There is no further response.
- The input ”Reference point reached” is set permanantly to 1 for fictitious axes.
- The PLC inputs ”Travel commands +/-” are assigned for all axes of the transformation grouping.

- For fictitious axes the exact stop signals are set alternately to the travel commands.
- The input ”Axis under postition control” is cancelled by cancelling the signal ”Servo enable” for a fictitious axis. There is no further response.
- If the PLC output ”Parking axis” is set for a real axis of the transformation grouping or ”Servo enable” is cancelled, alarm 168* ”Servo enable traversing axis” is output if a fictitious axis is programmed with an active transformation.
- If a real axis in the transformation grouping is assigned with ”Feed hold”, this applies to the entire grouping.
- If a real axis in the transformation grouping is switched to control from the follow-up mode, an inversed transformation of the fictitious coordinates occurs. Fictitious axes are not updated in follow-up mode.
- DRF
  Precondition for DRF in the fictitious system:
  The transformation must have been selected before.
- The program and thus the transformation are not stopped if the axis disable signal has been set for a real axis of the transformation grouping.
  This results in an offset between transformation and position control which can only be eliminated via RESET.
- When ”Clear residual path” occurs, the fictitious residual paths are cleared. The transformation causes the real axes of the transformation grouping to remain stationary.
- The signal ”Transformation active” is set in the channel-specific interface NC PLC for channel 1, if the transformation is active (see Appendix).
- The transformation can neither be selected nor cancelled via the PLC.

10.12.6 Explanation of the programming and operation of coordinate transformation

- Fictitious axes must not be programmed in the reset position (G130) alarm 2043.
- A transformation can only be activated from a reset position.
- Coordinate transformation is selected via the G function G131:
  The selection block must not contain any travel movements, miscellaneous functions, etc.
- No real axes or the transformation grouping must be programmed when transformation is active alarm 2043.
- Each selection/cancellation of transformation is connected to the function ”Clear buffer” (@714) . The @714 need not be programmed as it is automatically initiated by the control.
- Cutter radius compensation/tool nose radius compensation must be deselected before activating/cancelling the transformation (via @714) alarm 2081.
- Cutter radius compensation can be programmed when TRANSMIT is selected.

- The plane definition specified in the channel-specific machine data applies to the real system. When a transformation is selected, a plane is set for the **fictitious** system. The fictitious plane is defined in the transformation data by assigning the fictitious axes. Deviation from the basic plane can be explicitly programmed via G16. For TRANSMIT, the basic plane position is defined as G17 (A_{1F}-A_{2F}). G17 is automatically adjusted when a transformation is selected. The plane which was effective before selecting the transformation is automatically restored after the transformation is cancelled.



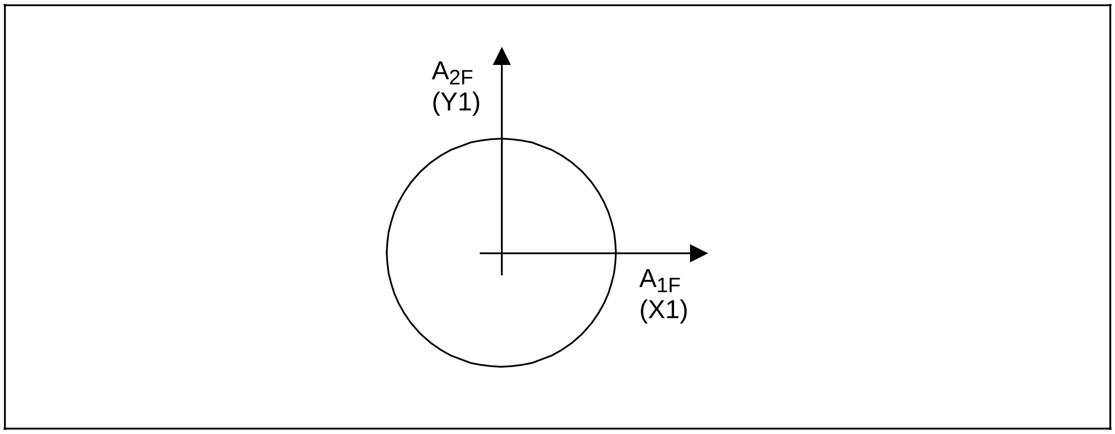
- If tools with length compensation L1 are used for TRANSMIT, the axis in which this compensation is to apply must be specified. The user can specify this axis with command G16 or in MD 5064 (name of infeed axis).
- A transformation must not be selected or cancelled within a contour block sequence.
- A block search to a program section where transformation is active is permitted.
- Block search (jump via @) to a program section where transformation is active is **not** permitted.
- PRESET offsets of real axes are ignored in the case of transformation.
- Fictitious axes can be traversed with the handwheel. Real axes can only be traversed with the handwheel when transformation is not active.
- DRF is possible in the case of fictitious axes, however, an error in speed calculation may occur.
- DRF is only possible in the case of real axes when the program block is active (not for NC STOP).
- The adjustable angle of rotation for coordinate system rotation (G54 to G57) must always be ZERO.
- The programmable angle of rotation for coordinate system rotation (G58, G59) must also be ZERO when transformation is selected or deselected.

- It should be noted that revolutional feedrate is frequently used on turning machines. When TRANSMIT is selected, revolutional feedrate must be deselected with G94 if necessary.
- REPOS offsets and the set/actual difference are only displayed in the axes of the current fictitious or real coordinate system.
- – Alarm 2035 may be activated in the case of small distances from the turning centre.
  – Possible causes: 1. Feedrate override greater than 100 %
  2. Quantisation error
  – Consequences: No NC stop.
  Form error at the contour because the rotary axis speed is limited to the maximum value.
  – Remedy: 1. Application of the largest possible cutter radius
  2. Reduction of the programmed feedrate
  3. Reduction of feedrate override

10.12.7 Examples of coordinate transformation TRANSMIT

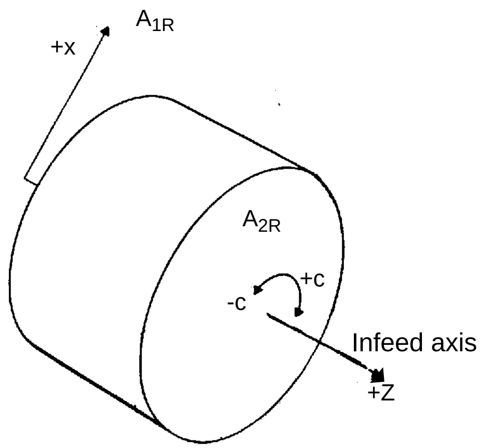
A transformation data record for the TRANSMIT transformation must be defined as follows:

NC MD 5061	G function for transformation selection	G131	1111 0001
NC MD 5062	Axis name 1st fictitious axis (A_{1F})		
NC MD 5063	Axis name 2nd fictitious axis (A_{2F})		



NC MD 5064 Axis name of infeed axis (real axis) Example: 0000 0010 (Z)

NC MD 5065 Axis name 1st real axis (A_{1R}) – linear axis
NC MD 5066 Axis name 2nd real axis (A_{2R}) – rotary axis



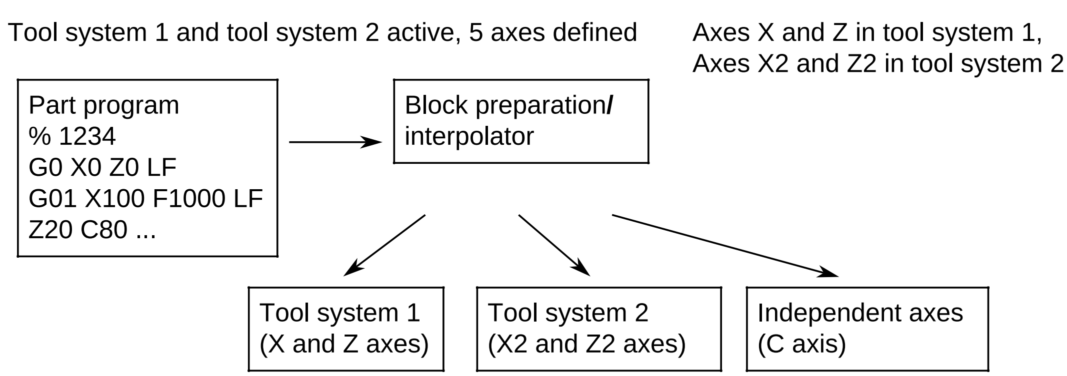
NC MD 5067 to 5069 unassigned (input: 1111 1111).

See Programming Guide for programming example.

10.13 Axis duplication

The ”Axis duplication” function allows the simultaneous machining of two workpieces or the complete machining of one workpiece by using one part program in one NC channel. The part program only contains the machining operations for one workpiece.

For every workpiece (for every workpiece side in the case of complete machining) seperate tool length compensations are allowed for, i. e. two different tool systems are required.



Example showing axis assignment

The two tool systems can be activated via the PLC NC interface. Axis duplication can be activated in the ”Automatic” and ”MDI” modes and is also possible with ”AUTO interrupted”.

10.13.1 Corresponding data

- MD 576* bits 5, 6, 7 — Axis number of synchronous axis, tool system 2
- MD 572*.0 — Also mirror zero offsets in the case of the ”Mirror axis” function
- MD 1096* — Values for the fictitious 2nd reference points

When the axis duplication option is set, the following machine data must be identical for both axes:

MD	344*	Modulo value for rotary axes
MD bit	560*.2	Rounding full/half degrees
MD bit	560*.3	Rounding for rotary axes
MD bit	560*.5	Software limit switches active
MD bit	560*.7	Actual value display modulo 360 degrees
MD bit	564*.5	Rotary axis
MD bit	572*.1	Transverse axis
MD bit	572*.2	Program modulo 360 degrees
MD bit	572*.3	Mirror tool offset for transverse axis
MD bit	572*.4	Rotary axis full/semicircle programming

Axis duplication OPTION

PLC signals	Q 81.2	Activate tool system 1
	Q 81.3	Activate tool system 2
	I 119.7	Axis 1 in duplication
	I 121.7	Axis 2 in duplication
	I 123.7	Axis 3 in duplication
	I 125.7	Axis 4 in duplication
	I 127.7	Axis 5 in duplication
	I 87.7	Axis 6 in duplication
	I 89.7	Axis 7 in duplication

Alarm	40	Incorrect data in MD 576*
Alarm	2192	Axis duplication active
Alarm	3000	General decoding error
Alarm	3003	Illegal address

10.13.2 Mode of operation

10.13.2.1 PLC/NC interface

The ”Axis duplication” function is controlled by the PLC program with the PLC signals Q 81.2 and Q81.3. This is only possible if the ”Axis duplication” function has been activated by one of our service engineers. The interface signals have a preparatory function. The required function is not activated in ”AUTOMATIC” or ”MDI” mode until NC start has been actuated .The activated function is modal until the next ”Reset” (Reset key or mode change Reset). The preselected function must not be changed during program processing (alarm message).

10.13.2.2 Mode functionality when the ”Axis duplication” function is active

PRESET and reference point approach

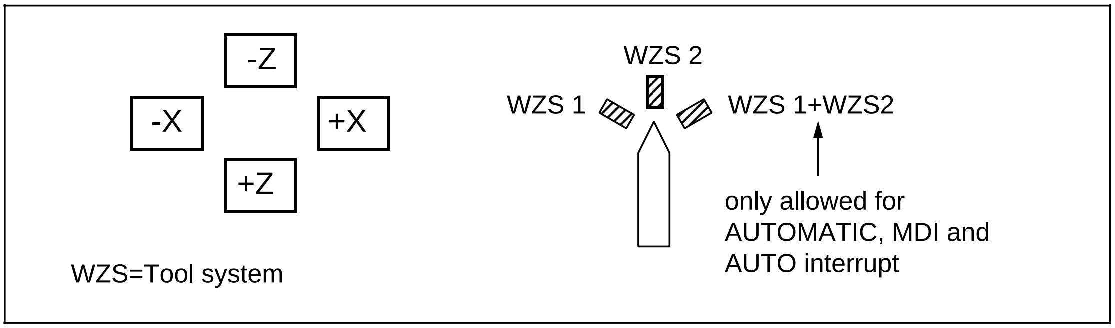
PRESET and reference point approach are the same as without axis duplication.

In other words, the axes must be operated individually. If tool system 2 does not have its own travel keys, these keys must be simulated in the PLC program.

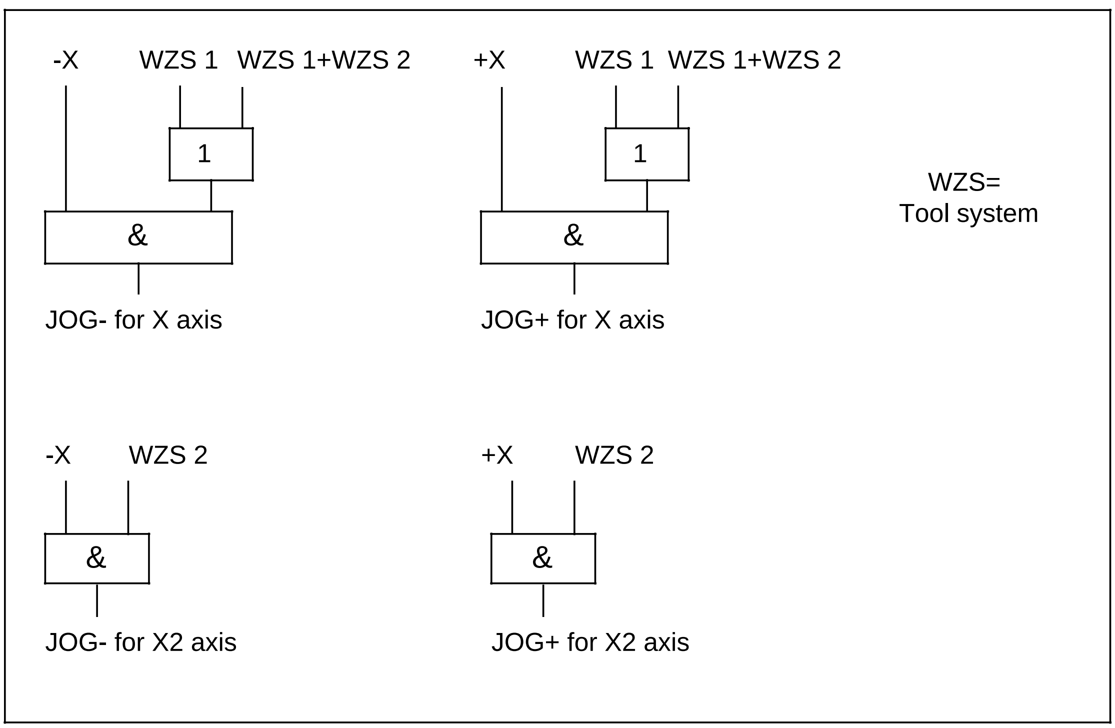
JOG, REPOS and INC without Automatic interrupt and not after block search

These operating modes are the same as without axis duplication. Tool system 1 **or** tool system 2 must carry 1-signal. This means that the axes must be attended to individually. If tool system 2 does not have its own travel keys, these keys must be simulated in the PLC program.

Example for keys:



Possible logic in the PLC program in the case of the X, Y axes for tool system 1 and X2, Y2 for tool system 2:



10.13.2.3 Automatic and MDI

The preselected function is stored with NC start. Preselection is effected via Q81.2 (tool system 1) or Q81.3 (tool system 2) and must not be changed until the next program end or RESET. Only the axes of tool system 1 can be programmed in the part program. Tool system 1, tool system 2 or tool system 1+tool system 2 are traversed in accordance with the signals Q81.2, Q81.3. The working area limits G25/G26 are duplicated.

Q 81.2	Q 81.3	Tool system 1 is traversed	Tool system 2 is traversed
1	0	X	-
0	1	-	X
1	1	X	X
0	0	not allowed when option is active	

The preselected function is modal until the next RESET or program end. This means that the axes can be traversed in the JOG, INC modes according to the preselected function even with AUTOMATIC interrupted.

When axis duplication is active, the axes can be traversed in different ways in REPOS mode after block search or AUTO interrupted:

Case 1: The axis of tool system 1 and the axis of tool system 2 have identical offsets. In this case, both axes (tool system 1, tool system 2) traverse the offset simultaneously after the travel key for the axis of tool system 1 has been actuated.

Case 2: The axis of tool system 1 and the axis of tool system 2 have different offsets. In this case, to traverse the offset, a travel key must be actuated for each axis.

Case 3: When NC start is selected, the offsets of all axes are traversed simultaneously.

10.13.2.4 Allowance for compensations when axis duplication is active

Zero offsets

The zero offsets G54 to G57 and the external zero offset remain axially active as before. Zero offsets G58 and G59 are assigned to the preselected axes (tool system1, tool system 2, or tool system 1+tool system 2 depending on the relevant preselection, tool system 1+tool system 2 for double axis when the axis is programmed). In the ”programmed zero offsets” display, the zero offset is always displayed for both axes (programmed and duplicated axis) irrespective of the active signals Q 81.2/Q 81.3.

If the machine data bit 572*.0=1, zero offsets are also mirrorred (for complete machining) when the PLC signal ”Mirror” is active.

Scale factors

The SD option bits of tool system 1 act as scale factors.

Tool offsets

The tool offset memories D1 ... D49 are permanently assigned to tool system1. Tool offset memories D51 ... D91 are permanently assigned to tool system 2. Only the tool offset memory of tool system 1 is programmed. The associated tool offset memory is then activated for tool system 2 with an offset of 50 with only the tool lengths being relevant.

Example: Tool system 1 D3
Tool system 2 D53

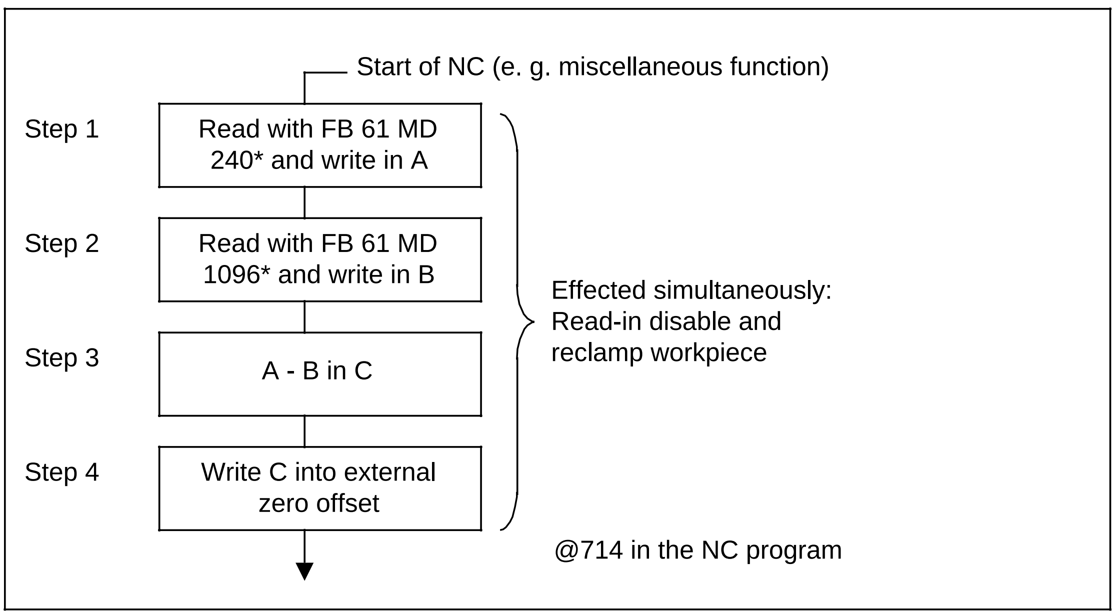
The tool radius must be identical for the tool pairs,

the tool lengths may differ.

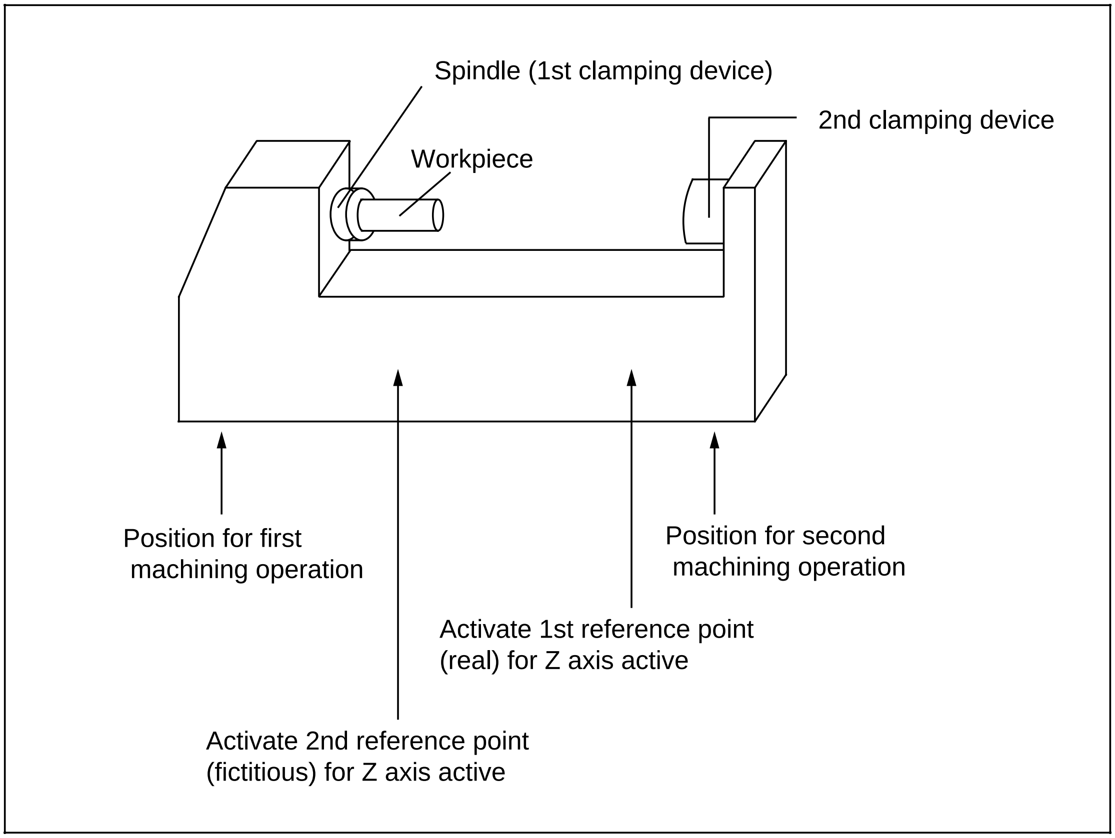
10.13.2.5 Complete machining (special function)

For the complete machining of a workpiece, machine data (MD 1096*) are provided for the fictitious second reference points in addition to the machine data for mirroring the zero offsets (MD 572*.0=1). The difference between the machine data MD 240* (reference point value) and MD 1096* must be specified in the PLC as external zero offset. The SINUMERIK is thus given the illusion of a second reference point. The following part of the NC program must then refer to this fictitious second reference point.

Example showing the sequence in the PLC program:



Application example for complete machining on a turning machine



Example for complete machining

10.13.3 Basic function conditions

- One measuring circuit must be provided for each axis for axis duplication. Consequently, in the SINUMERIK 810/820 GA3 a maximum of 2 axes of the possible 5 axes can be duplicated. The 5th axis will then still be available for one independent axis.
- The duplicated axes have their own tool system. The leading axes are valid for tool system 1.
  The duplicated axes are valid for tool system 2. The tool offset memories D1 ... D49 are valid for tool system 1. The tool offset memories D51 ... D99 are valid for tool system 2.
- In axis duplication, the programmed set velocities apply to each tool system.
- The handwheels act directly on the specified axes.
- The software limit switches act on each axis separately, i. e. a separate software limit switch range must be entered in the machine data for each axis in the case of axis duplication.
- Block search is not subject to any restriction.
- The ”Soft approach/retraction” function can be used.
- The ”Coordinate rotation” function can be used.
- The ”Spline interpolation” function can be used.
- Contour definitions are applicable.
- When duplication is active, unequal PLC interface signals for associated axes of tool system 1 and tool system 2 (Mirror, Feedrate enable, Servo enable, Follow-up mode, Axis disable, Parking) cause alarm 2190 (”Axis duplication active”) to be triggered. In other words, the PLC interface signals must be identical for associated axes of tool system 1 and tool system 2.
- Except for @440 (programmed axis position) all axis-related @ act just as before.
- With @720 (inprocess measurement) the programmed traversing path is duplicated to the relevant axis but only one actual value is measured. Both axes stop moving when the sensor is triggered.
- As regards tool offsets, it should be noted that the tool offset memories D1 ... D49 are valid for tool system 1. Tool offset memories D51 ... D99 are valid for tool system 2. Only the tool offset memory of tool system 1 is called in the program. The tool in tool system 2 must have the following tool offset memories:
  $D_{wzs\ 2}=D_{wzs\ 1}+50$.
  The tool radius must be identical for the two tools ($D_{wzs\ 2}=D_{wzs\ 1}+50$), length compensations may differ. The tool type in D51 to D99 for tool system 2 is ignored and can be assigned zero.
  When axis duplication is active and MD5011 bit 3=1 or bit 4=1, length P2 and length compensation P5 are calculated in the diameter for tool type P1=0.

Axis duplication can only be run in channel 1.

Possible collision must be taken into account for measurement in the case of axis duplication.

10.13.4 Examples

Three examples are given to illustrate the mode of functioning of axis duplication with the same machine configuration.

The axis duplication function is active. Machine configuration (defined NC machine data):

MD 5680	Axis name 1st axis=X2
MD 5681	Axis name 2nd axis=Z
MD 5682	Axis name 3th axis=C (rotary axis)
MD 5683	Axis name 4th axis=Z2
MD 5684	Axis name 5th axis=X1
MD 5760 bits 7, 6, 5	= 0
MD 5761 bits 7, 6, 5	= 100 binary (Z axis is duplicated to Z2 axis)
MD 5762 bits 7, 6, 5	= 0
MD 5763 bits 7, 6, 5	= 0
MD 5764 bits 7, 6, 5	= 001 binary (X1 axis is duplicated to X2 axis)

Example 1: PLC signal Q 81.2=1
PLC signal Q 81.3=0

Mode of functioning when tool system 1 is active:

Part program	Effect
N0000 G0 X1=0 Z0 D5 LF	Axes X1 and Z traverse to 0 in rapid mode, the selected tool is D5
N0010 G01 X1=100 F 2000 LF	Axis X1 traverses to 100 at a speed of 2000
N0020 Z 30 C 50 LF	Axes Z and C (rotary axis) interpolate with speed 2000 to Z30 C50
N0030 G33 X1=50 I 2 LF	Thread cutting with X1
N0040 ...	

Example 2: PLC signal Q 81.2=0
PLC signal Q 81.3=1

Mode of functioning when tool system 2 is active:

Part program	Effect
N0000 G0 X1=0 Z0 LF	Axes X2 and Z2 traverse to 0 in rapid mode, the selected tool is D55
N0010 G01 X1=100 F 2000 LF	Axis X2 traverses to 100 at speed 2000
N0020 Z 30 C 50 LF	Axes Z2 and C (rotary axis) interpolate with speed 2000 to Z30 C50
N0030 G33 X1=50 I 2 LF	Thread cutting with X2
N0040 ...	

Example 3: PLC signal Q 81.2=1
PLC signal Q 81.3=1

Mode of functioning when tool systems 1 and 2 are active (axis duplication):

Part program	Effect
N0000 G0 X1=0 Z0 LF	Axes X1/X2 and Z1/Z2 traverse to 0 in rapid mode, the selected tool is D5 for tool system 1 and D55 for tool system 2.
N0010 G01 X1=100 F 2000 LF	Axes X1/X2 traverse to 100 at speed 2000
N0020 Z 30 C 50 LF	Axes Z/Z2 and C (rotary axis) interpolate with speed 2000 to Z30 C50
N0030 G33 X1=50 I 2 LF	Thread cutting with axes X1/X2
N0040 ...	

10.14 Software cams

10.14.1 Function description

Cam signals

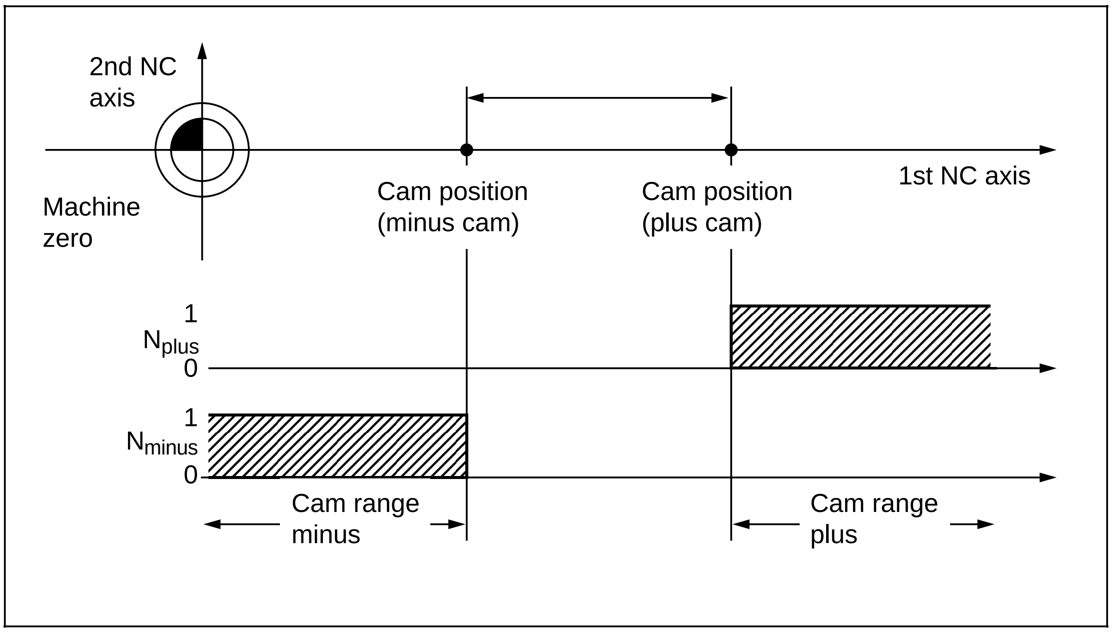
Cam signals are NC control signals. They simulate an operating cam of infinite length which is activated in a defined approach direction at a defined position (cam position).

The bit in DB 39 in the PLC assigned to the cam is set when the relevant cam area is reached and cancelled on exit of the cam area.

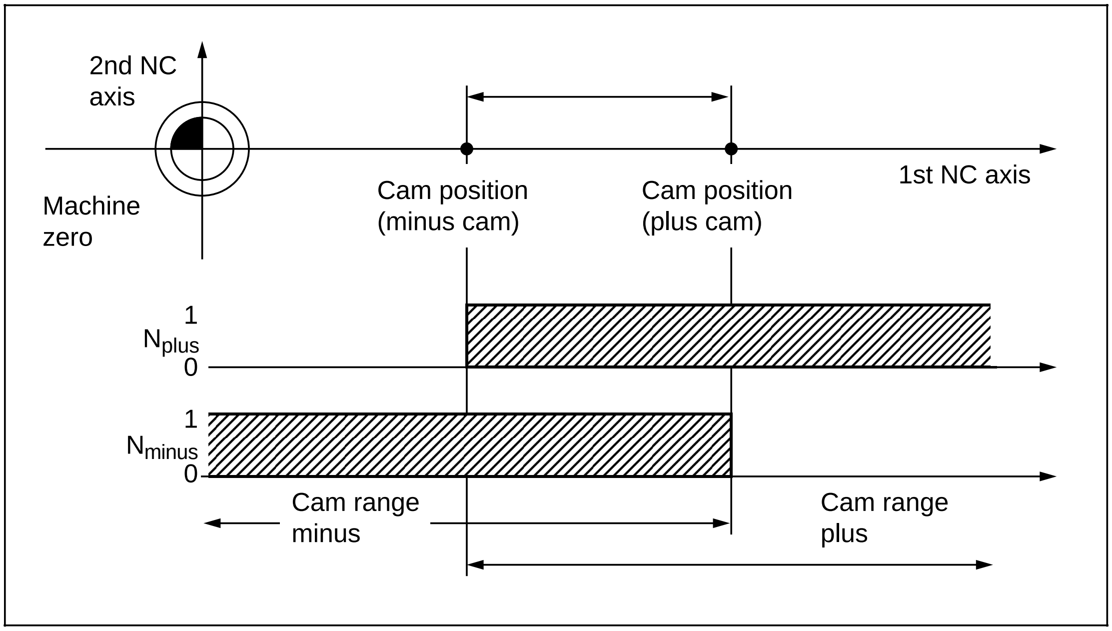
The cam signals are not output until reference point approach has been performed in the axes.

Cam pair and cam area

One cam pair consists of one plus cam and one minus cam. The plus cam is assigned the axis range which is greater than its cam position while the minus cam is assigned the axis which is less than its cam position. The axis range assigned to the cam is termed cam range .



Minus cam<plus cam



Plus cam<minus cam

Cam parameters

All cam parameters are contained in one R parameter block. The R parameter block is termed cam parameter block and contains the positions of ten cams which are grouped as five cam pairs. The cam positions refer to machine zero and are used in the unit system active for the machine (metric or inch). The cam positions are not checked for maximum position values (maximum traversing range).

The start of the cam parameter block (Rxxx) is defined via DW 5 in DB 39.

Position minus cam 1	R xxx	Cam pair 1	Cam parameter block
Position plus cam 1	R xxx+1		
Position minus cam 2	R xxx+2	Cam pair 2	
Position plus cam 2	R xxx+3		
Position minus cam 3	R xxx+4	Cam pair 3	
Position plus cam 3	R xxx+5		
Position minus cam 4	R xxx+6	Cam pair 4	
Position plus cam 4	R xxx+7		
Position minus cam 5	R xxx+8	Cam pair 5	
Position plus cam 5	R xxx+9		

10.14.2 Activation

Activation of function

The software cam function is an option.

DB 39 is set up in the PLC on cold restart if the software cam option is activated.

The cam signals are only processed when the option is active and they are only updated in DB 39 of the PLC prior to starting a new PLC cycle when they are changed.

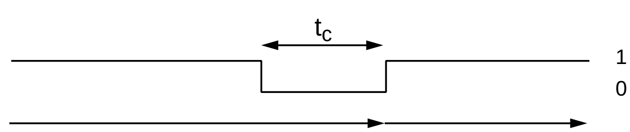
The cam signals are not output until reference point approach has been performed in the axes. The exception to this rule are axes for which NC MD 560* bit 4 (no start disable prior to reference point approch) is set. Cam signals are not generated until cam parameters and axis assignments have been activated via DB 39.

Activation of cam parameters

Cam parameters are stored in R parameters in pairs (plus/minus value). All cam parameters together form one parameter block. When the relevant bits are set in DB 39 some or all parameter pairs are updated for value transfer by the NC.

The associated parameter pairs are read out from the parameter block by the NC only as a result of signal change of the bits from 0 to 1 (bits must be greater or equal to 1 PLC cycle 0). A valid start number must also be entered for the R parameters in DB 39 at the instant of the 0/1 edge to fix the start of the parameter block within the R parameters.

The values of a cam pair can be declared invalid by not assigning the cam pair to an axis.



Bit in DB 39 for activating all/one cam pair(s)
t_c: PLC cycle time

Those cam values of all/one cam pair(s) are active which have been transferred prior to the change.

The new values from the cam parameter block of all/one cam pair(s) are active.

Axis assignment of cam parameters

In the cam parameter block cam values are stored with which the positions of the required axes are compared. Axis/cam pair assignment is effected in DB 39 via the relevant bits.

Several cam pairs can be assigned to each axis but the same cam pairs must not be assigned to several axes at the same time.

The axis assignments in DB 39 are taken over by the NC if axis assignments have changed in DB 39.

Note: The software cam function is only permissible for linear axes.

Example: DB 39

DL DR

DW 1:	0000 0001	0000 0010	i. e. axis 1 with cam pair 1, axis 2 with cam pair 2
DW 2:	0000 1100	0000 0000	i. e. axis 3 with cam pairs 3 and 4

10.14.3 Mode of operation in the individual operating modes

Automatic, MDI

When the software cam function is active, the cam signals are processed and output in Reset state with AUTOMATIC stop, AUTOMATIC active.

JOG, INC, REPOS

When the software cam function is active, the cam signals are processed and output in Reset state or with AUTOMATIC interrupted.

PRESET, REF

The software cams are not active in this operating mode.

10.14.4 R parameters

The values of the 5 software cams are stored in 5 consecutive R parameter pairs. For each software cam there is one R parameter for negative cam direction and one for positive cam direction. The number of the first R parameter of this parameter block is specified in DB 39 via the PLC.

A cam parameter block can only be formed from the global R parameters.

10.14.5 PLC interface

The following are specified in DB 39 via the PLC:

- Which cams are assigned to which axes?
- When are the cam values in the R parameters valid?
- Where does the cam parameter start?

The cam signals determined by the NC are also stored in DB 39, divided into 2 bits each for the plus and minus range of each cam pair.

DB 39

	Byte no.		Bit 7	6	5	4	3	2	1	0
To NC	DL O Strobes for value transfer of the cam parameter values									All values
To NC	DR O					Cam pair 5	4	3	2	1
To NC	Assignment to axis DL 1	1				5	4	3	2	1
To NC	DR 1	2				5	4	3	2	1
To NC	DL 2	3				5	4	3	2	1
To NC	DR 2	4				5	4	3	2	1
To NC	DL 3	5				5	4	3	2	1
To NC	DR 3	6				5	4	1) 3	2	1
To NC	DL 4	7				5	4	1) 3	2	1
To NC	DR 4					Reserved				
To NC	DL 5 DR 5		Start of cam parameter block (R no. for 1st value, KF format)							
From NC	DL 6 Cam signals active								Cam pair 5 +	-
From NC	DR 6		Cam pair 4 +	-	Cam pair 3 +	-	Cam pair 2 +	-	Cam pair 1 +	-

1) *These axis assignments can only be used in conjunction with the Transmit option (for axes 6 and 7).*

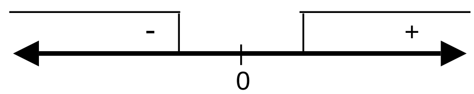
Example:

	DB 39:	
DL 0:	KB00000000	
DR 0:	KB00010111	Cam values for cams 1, 2, 3 and 5 active
DL 1:	KB00000010	Axis 1 with cam pair 2
DR 1:	KB00000001	Axis 2 with cam pair 1
DL 2:	KB00010100	Axis 3 with cam pairs 3 and 5
DR 2:	KB00000000	Axis 4 without cam pair
DL 3:	KB00000000	Axis 5 without cam pair
DW 5:	KF950	Start of cam parameter block from R950 to R959 inclusive
DL 6:	KBxxxxxxxx	Cam signals
DR 6:	KBxxxxxxxx	Cam signals

R parameters

R 950 = -132000
R 951 = +140000

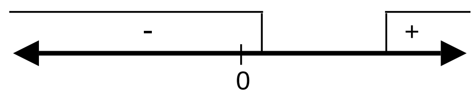
Cam pair 1



Axis 2

R 952 = +20000
R 953 = +200000

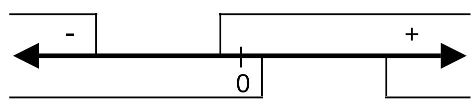
Cam pair 2



Axis 1

R 954 = -300000
R 955 = -200000

Cam pair 3
Cam pair 5



Axis 3

R 956 = 0
R 957 = 0

R 958 = +15000
R 959 = +250000

10.15 Menu insertions by the user

See Interface Description Part 1 (signals)

for a detailed description of this function.

It is possible for the user to interrupt the existing menu structure with

1. a new interface (DB 40) in the PLC program as for SINUMERIK 850/880
2. a new softkey function (softkey 56) as for SINUMERIK 850/880

Only menu blocks of the system and user data area can be inserted. No insertions are possible in setup mode.

Large actual value display

The "Large actual value display" can be selected with every menu just as before. If a new menu block is activated with a new display, the "Large actual value display" disappears as before.

Problems involved in insertion

Menus can now be inserted practically any time at any place in the normal operation tree. To give an example, menus can be inserted in AUTO mode that normally are only to be found in REPOS mode etc. The predetermined tree structure can be skipped arbitrarily. This involves the risk that menus are selected which normally can only be reached via a softkey function. This may lead to malfunctions with serious consequences because the relevant softkey function has not been executed. It is the responsibility of the PLC and the UMS to avoid this. No monitoring is provided by the system program.

Teach-in/Playback

On cancellation of Teach-in/Playback, the operator is taken back to the basic menu of the operating mode.

Recall

A specified softkey function is not executed if Recall is initiated in an entry menu.

Recall to the basic menu of the operating menu (Shift+Recall)

The operator is taken back to the basic menu of the operating mode by pressing Shift+Recall. If insertions were active in this operating mode, the menu stack is cleared and "Special Recall" is deselected without the PLC priority being affected.

Funktionsübersicht

Function	PLC	NC	SK (Softkey)
Insert system menu	x	x	x
Insert UMS menu	x	x	x
Specify new XM in the system	x		
Specify new XM in the UMS	x		
Superimpose PLC menu with priority	x		x
Execute Recall		x	
No basic display with AUTO/JOG	x		
Set priority for superimposing displays	x		
Execute channel change	x		
Request Special Recall	x		
Initiate ”Menu Stack Reset”	x		
Input disable for Input/Edit/Cancel	x		
Display keycode from NC	x		
Transfer keycode to NC	x		
Select basic menu of operating mode	with Shift+Recall via operator panel		

Interface signals

Byte no.	Bit 15 / 7	14 / 6	13 / 5	12 / 4	11 / 3	10 / 2	9 / 1	8 / 0
DL 0								
⋮								
DL 7	Display: Keycode from NC							
⋮								
DL 42					X	INSTAL-LATION	DISPLAY END	APPLICA-TION
DR 42	Display: Menu number							
⋮								
DL 51								Start
DR 51	X	IB active				X	Error	o.k.
DL 52	Job number (KF) (1 ... 14)							
DR 52								

X Bits, bytes or words assigned with X must not be used.

Interface signals

Byte no.	Bit no. 15 / 7	14 / 6	13 / 5	12 / 4	11 / 3	10 / 2	9 / 1	8 / 0
DL 53								
DR 53	Menu number (KF) (1 ... 254)							
DL 54	X							
DR 54	Channel number 0 or 1 ... 3							
DL 55	Inhibit	Strobe key				INPUT	EDIT	CANCEL
DR 55	Key code to NC							
DL 56								Spec. RECALL
DR 56	Display: Channel number							
DL 57								
. . .								
DR 63								

X Bits, bytes or words assigned with X must not be used.

10.16 Indirect addressing

Indirect addressing is a function of the NC and can be used with the help of the NC workstation WS 800A Version 3.0.

This function provides the user with an additional degree of freedom when configuring displays.

The basic principle is that specific information can be moved from the display description into PLC memory cells that the user can access (PLC program or status). This information can therefore be modified without having to modify the display description itself.

The PLC memory cells correspond to data words in data blocks.

The function ”Indirect addressing”can be configured only in the display type ”Extended screen form (EE)”. For configuring the display type ”EE” on the WS800A (SW3 and higher) the control type SINUMERIK 880N must be selected at the present time and the following restrictions must be observed by the user. With the preselected control type SINUMERIK 880N, the PLC programs of the SINUMERIK 810/820 can also be integrated on the WS800A.

In this display type ”EE”, the display elements ”Indirect element (IE)” must be specified for indirect addressing. A distinction is made in this display element ”IE” between direct addressing and indirect addressing. In the display description, a basic data group must be displayed for this (see WS800A documentation).

The following restrictions must be observed compared with the WS800A documentation for display elements ”IE” for the SINUMERIK 810/820 GA3. The SINUMERIK 810/820 GA3 itself does not check whether these restrictions have been observed.

In **direct addressing** the following data groups are allowed:

Designation (System software)	Data group
R parameter	19H
Actual values	1AH
Feedrate	1CH
Input buffer	1DH
Spindle channel-specific	1FH
Override	20H
Variable address driver (var. axis names only)	42H
Auxiliary functions (BAM)	70H
PLC inputs	82H
PLC outputs	83H
Flags	84H
Timers	85H
Counters	86H
Data words	87H
PLC inputs (status)	89H
PLC outputs (status)	8AH
Flags(status)	8BH
Timers (status)	8CH
Counters (status)	8DH
Data words (status)	8EH

In **indirect addressing** the SINUMERIK 810/820 GA3 evaluates the element type only for the basic data group 135 (data words, 87H):

- **Element type 0 ”Value”**
With this type, the contents of the configured memory cell are represented in the display format that is also configured.
Information in the display description (BB) for the basic data group:

DG:	87H (=135)
D.type:	0 - 15/16,17/80,81/32,33 (bit No./DL/DR/DW)
DB No.:	0 - 255
D.No.:	0 - DB length
Field format:	as described in WS800A documentation
Field information:	as described in WS800A documentation

• **Element type 2 ”UMS text No.”**
With this type, a memory cell is configured containing a number that is assigned to a text stored in the UMS. Each text number must be specified as a fixed-point number. A user text must be available in the UMS for this purpose.
Information in the display description (BB) for the basic data group:

DG:	87H (=135)
D.type:	16,17/80,81/32,33 (DL/DR/DW)
DB No.:	0 - 255
D.No.:	0 - DB length
Field format:	21H
Field information:	01 - 41 = text length = number of characters

• **Element type 3 ”Decision bit UMS text”**
With this type, one of two text numbers is selected with the help of a PLC bit address. The text numbers are given in the display description and refer to a UMS text. A user text must be available in the UMS for this purpose.
Information in the display description (BB) for the basic data group:

DG:	87H (=135)
D.type:	0 - 15 (bit no.)
DB No.:	0 - 255
D.No.:	0 - DB length
Field format:	21H
Field information:	01 - 41 = text length = number of characters

10.17 Service displays

For drive optimization and fault diagnosis purposes, data from and to the axis must be accessed and displayed.

Selection of service data:

- ”DIAGNOSIS” softkey
- Key $\boxed{>}$
- ”SERVICE AXIS” softkey”

or

- ”SERVICE SPINDLE” softkey

10.17.1 Service axes

The ”Page” key makes it possible to switch the display to the required axis.

Display:

Axis 1	1st axis
Axis 2	2nd axis
Axis 3	3rd axis
Axis 4	4th axis
Axis 5	5th axis
Axis 6	6th axis
Axis 7	7th axis

On the SINUMERIK 810 GA3/820 GA3, the following data is displayed:

- Following error — Reference between set position and absolute actual position. The following error is displayed in units of position control resolution.

  Example: If 200 000 is displayed this means that with a position control resolution of $0.5 \cdot 10^{-3}$ mm there is a following error of 1 mm.

- Absolute actual value — The actual position of the axis. The position is displayed in the machine coordinate system (ZO and TO not taken into account) in units of position control

  Example: When 200 000 is displayed this means that with the position control resolution of $0.5 \cdot 10^{-3}$ mm the axis is at position 100 mm with reference to the machine zero.

- Setpoint value — The target position as programmed. In the target position (zero speed) the set position is equal to the absolute actual position (should there be any difference this can be removed by drift compensation). The set position is displayed in units of position control resolutions.

  Example: If 202 000 is displayed this means that with a position control resolution of $0.5 \cdot 10^{-3}$ mm the set position is 101 mm with reference to the machine zero.

- Set speed — The set speed determined by the control which is fed to the drive actuator as an analog voltage value.
  Unit: 1.22 mV ($\hat{=}$ 1 VELO)
  8192 Velo $\hat{=}$ 10V

  Example: A displayed value of 5638 corresponds to a set speed of 6.87836 V

- Actual value difference: — The pulses coming from the measuring systems x 4 per position control scan time (5 ms). Multiplication by 4 is necessary so that the actual value difference (scan time
  5 ms) can be compared with the setpoint difference (scan time 20 ms).
  Unit: Position control resolution (standard: 0.5µm)

  Example: A displayed value of 24 corresponds to a distance of 12 µm per 20 ms with a position control resolution of 0.5µm.

- Setpoint difference — The pulses output by the interpolator to the position control per interpolator frequency period (20 ms).
  Unit: Position control resolution (standard 0.5µm)

  Example: A displayed value of 18 corresponds to a distance of 9µm per 20 ms with a position control resolution of 0.5 µm.

- Contour deviation — Actual contour deviation (variations in following error caused by compensating processes at the speed controller due to load changes) or by the follow-on path of the infeed axis compared with the rotary axis when G36 is programmed.
  Unit: Position control resolution (standard: 0.5µm), for G36 input resolution

  Example: A displayed value of 2 corresponds to a contour deviation of 1µm with position control resolution 0.5µm.

- Status absolute module error number
- OFL — Overflow
  If the counter can no longer keep up with pulse rates which are too fast, 1 is set.
- By means of the softkey INTERF. SIGNALS, the statuses of important interface signals can be viewed for all defined axes.

10.17.2 Service data for spindle

With SINUMERIK 810 GA3/820 GA3 the following data is displayed:

- Set speed (VELO) — Set speed/analog voltage value determined by the control
- Actual speed (rev/min) — Actual speed of the spindle. The actual speed is displayed in rev/min.
- Actual position (degrees) — Display of the spindle position in degrees. The position is displayed within a range of 0.1 to 359.9 degrees.
- Set position (degrees) — Programmed target position. Display in the range 0.1...359.9 degrees
- Actual position (degrees) — Display of the spindle position in degrees. The position is displayed within a range of 0.1 to 359.9 degrees.
- Override (%)
- Set gear
- Actual gear

10.18 Channel structure of SINUMERIK 810 GA3/820 GA3

The control has been provided with 3 channels to cater for future SINUMERIK 810 GA2 machine requirements.

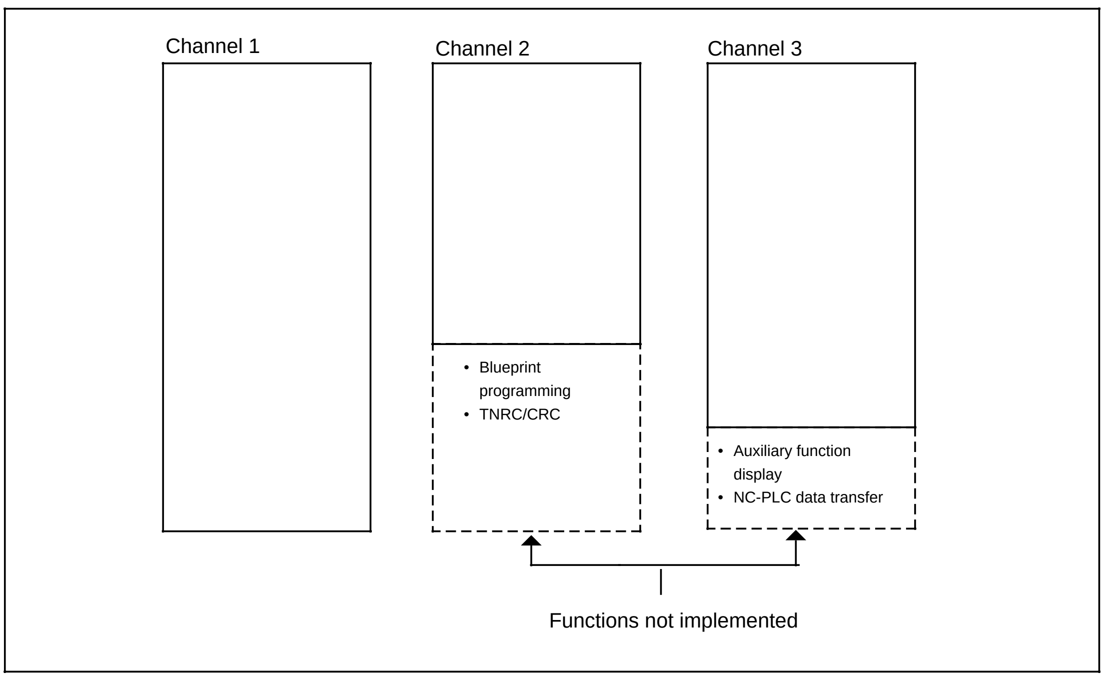
In addition to other structural operations such as program editing and interfacing at the same time as processing in automatic mode, these channels make it possible to process 2 different programs **simultaneously**.

The significance of the 3 channels is as follows:

Channel 1: Main channel for processing programs and spindle programming.
Channel 2: Auxiliary channel for processing programs for supplementary axes or for arithmetic functions in the background.
Channel 3: Graphics simulation to display programs on the screen.

All three channels can basically operate simultaneously. However, collision problems occur with a small number of functions (see end of Section 10.18).

Functional scope of channels:



The auxiliary channel (channel 2) has full capabilities, with the exception of those functions which are not yet available. Its primary function is to perform background calculations or to execute auxiliary movements (tool change, ...).

Block processing is separate in each channel, the channel-specific actual axis position being held internally. The channel-specific actual axis position is only updated in the channel in which the axis is traversed.
In other words: Without special measures movements in channel 1 are not taken into account in channel 2, and vice versa.
If synchronization of the channel-specific actual values is requested, the actual value can be taken from the previous channel with the NC STOP and NC START signals in the channel concerned.

PLC control of channels

The complete VDI interface between the NC and PLC is designed so that 2 channels can be controlled and influenced independently of each other.

NC PLC user interface

	Bit:	7	6	5	4	3	2	1	0	
Program commands	Channel 1	M00 / / / M01	M02 / / / M30	G33 / / / G63	G00	G96		Program interrupted	Program running	IB 102
	Channel 2	M00 / / / M01	M02 / / / M30	G33 / / / G63	G00	G96		Program interrupted	Program running	IB 108

All signals of importance for influencing of the two channels are available in duplicate. Only the operating mode is identical for both channels. The two channels can also be synchronized at specific points by means of the PLC using M functions and the ”Read-in enable” signal.

RESET (key) - Q 82.6- is not channel-dependent and thus acts simultaneously on all channels. NC Start only acts on channel 1.

With PLC MD 2002 bit 3 (transfer machine control panel from input to output image) only the signals for channel 1 are activated, thus preventing unintentional starting of channel 2.

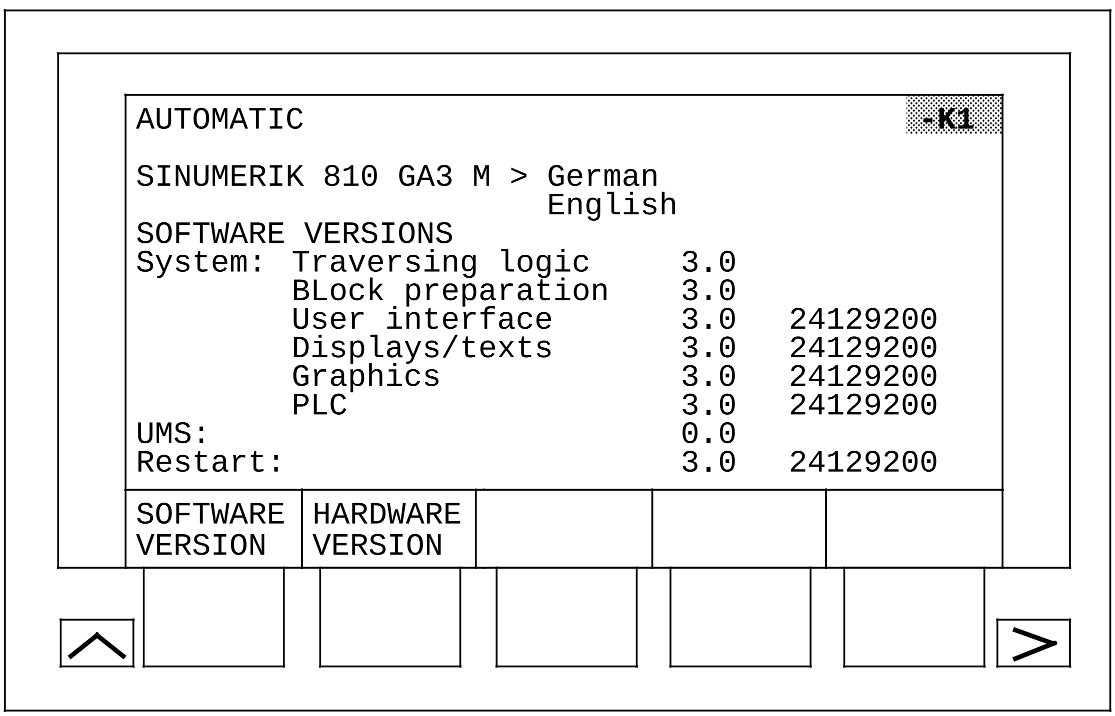
	Bit:	7	6	5	4	3	2	1	0	
Sub-modes	Channel 1	Skip block		DEC single block	Dry run feed-rate	M 01 active	Rap. trav. override active	DRF selected		IB 104
	Channel 2	Skip block		DEC single block	Dry run feed-rate	M 01 active	Rap. trav. override active	DRF selected		IB 110

PLC NC user interface

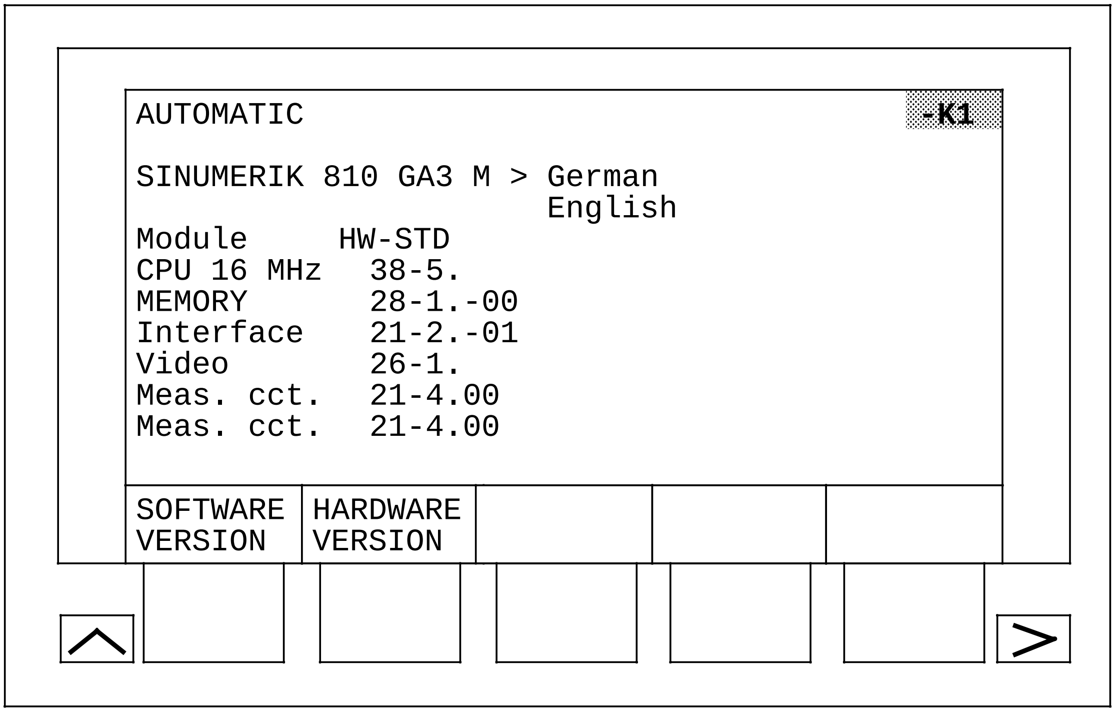
Bit	7	6	5	4	3	2	1	0	
	D R F active				Operating modes D	C	B	A	AB 82
Program influencing Channel 1	Skip block	Single block	DEC single block	Dry run feedrate	M01 active				AB 83
Program influencing Channel 1			Read-in enable		Clear distance to go	Clear repetitions sub-routine	NC Stop	NC Start	AB 87
Program influencing Channel 2	Skip block	Single block	DEC single block	Dry run feedrate	M01 active				AB 92
Program influencing Channel 2			Read-in enable		Clear distance to go	Clear repetitions sub-routine	NC Stop	NC Start	AB 96

10.19 Hardware and software version display

The diagnostics branch has been expanded to include the hardware and software version. In these displays the hardware number (Order no.) of the individual modules and the software version of the EPROM submodules can be read. The following display appears:



Software version display (example)



Hardware version display (example)

10.20 Integrated PLC auxiliary axes (Option N06)

Requirements:
- Option P06 or P08 ”PLC expansion unit”
- Option F72 ”External data input”
- WF 725/726 module

Mode of operation

The ”Integrated auxiliary axes” function allows the WF 725/726 module to be used in the expansion unit (EU) for the implementation of 1 to 3 auxiliary axes, e.g. to control a rotary table or a tool magazine.

Only 1 WF module can be connected. Data exchange between the WF module and the PLC or between the PLC and the NC is organized through PLC function blocks which are stored in a separate macro memory area. These function blocks do not load the PLC program memory. For them to function the control option F72 (external data transfer) is required. The FB package (”Basic information”) controls data exchange between WFs and PLCs. The FB package (”Programming functions”) transfers the part programs from the NC part program memory and the program setting via parameter assignment and data blocks to the WF module. Display of the actual values of the WF module is not implemented.

No further details are given here on the mode of operation of the function ”Integrated auxiliary axes” because there is a special Planning Guide on this subject.
Order number: E80850-J154-X-A1

11 Alarms

11.1 General

The control contains permanently active monitors which detect malfunctions in the NC, interface controller and machine at such an early stage that damage to the workpiece, tool or machine is largely ruled out.

In the event of malfunctions, machining is first interrupted and the drives shut down, the cause of the fault being stored and displayed as an alarm. At the same time the PLC is informed that an NC alarm is present.

Monitors exist for the following:

- Read-in
- Format
- Measuring-circuit cables
- Position encoder and drive
- Contour
- Spindle speed
- Enable signals
- Voltage
- Temperature
- Microprocessor
- Serial interfaces
- Data transfer between NC and PLC
- Condition of back-up battery
- System program memory
- User program memory

11.2 Display of all messages and alarms with DIAGNOSIS softkey

If the monitor responds, this may be due to a number of **different malfunctions simultaneously**.

However, only the **least significant alarm number** is displayed in the alarm line.

An overview of other alarms/messages is provided as follows:

Select **DIAGNOSIS** softkey function
in one of the 7 operating modes

↓

Select **NC ALARM**
or **PLC ALARM**
or **PLC MESSAGE**
softkey function

Exception: Overall reset commissioning mode

Basic display

Key >

Select **NC alarms** softkey

11.3 Alarm numbers and alarm groups / Clearing alarms

The alarms are brocken down into 7 alarm groups (5 NC alarm groups and 2 PLC alarm groups).

NC alarms:

- POWER ON alarms
- RS232C **(**V.24) alarms
- RESET alarms /axis-specific
- RESET alarms /general
- ERASE alarms

PLC alarms:

- PLC error messages
- PLC operational messages

The texts for the PLC error messages and PLC operational messages are stored in RAM on the CPU. They are input in the installation overall reset mode (see MD no. 5012).

Tabular overview with assignment of alarm number and clear mode:

Alarm number	Alarm group	Alarm cleared only by ...
1 .... 15 40 99	**POWER ON alarms**	Switching on the control
16 ... 39	**V.24 (RS 232) alarms**	1. Calling softkey menu containing the "DATA IN-OUT" function 1) 2. Actuating the "DATA IN-OUT" softkey 3. Actuating the "STOP" softkey
100* 196*	**RESET alarms/ axis-specific (*=axis no.)**	Actuating the RESET key
132*	**POWER ON alarms/ axis-specific (*=axis no.)**	Switching the control off/on
2000 2999	**RESET alarms/general**	Actuating the RESET key
3000 3087	**ERASE alarms**	Actuating the acknowledgement key
6000 .... 6063 6100 6163	**PLC user alarms PLC error message if no alarm 3 is present**	Actuating the acknowledgement key
7000 7063	**PLC operational message**	These messages are reset automatically by the PLC program

1) ***Additional note:***

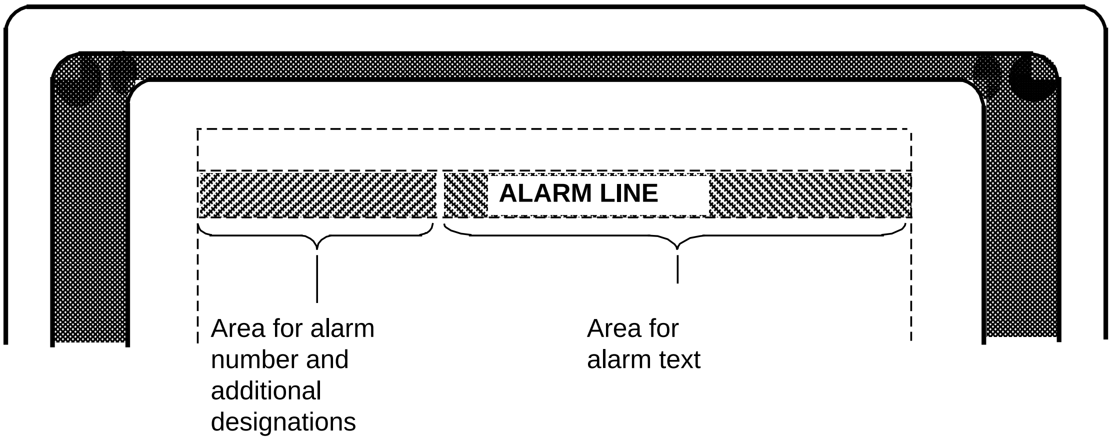
The"DATA IN-OUT" function can be called in the following modes:

- ***AUTOMATIC***
- ***JOG***
- ***REFPOINT***
- ***INC1 ... INC 10 000***
- ***PRESET***
- ***Installation mode***

11.4 Alarm display on screen

Messages from the monitor are displayed on the screen in the "Alarm line".

The "Alarm line" is the second line on the screen from the top.

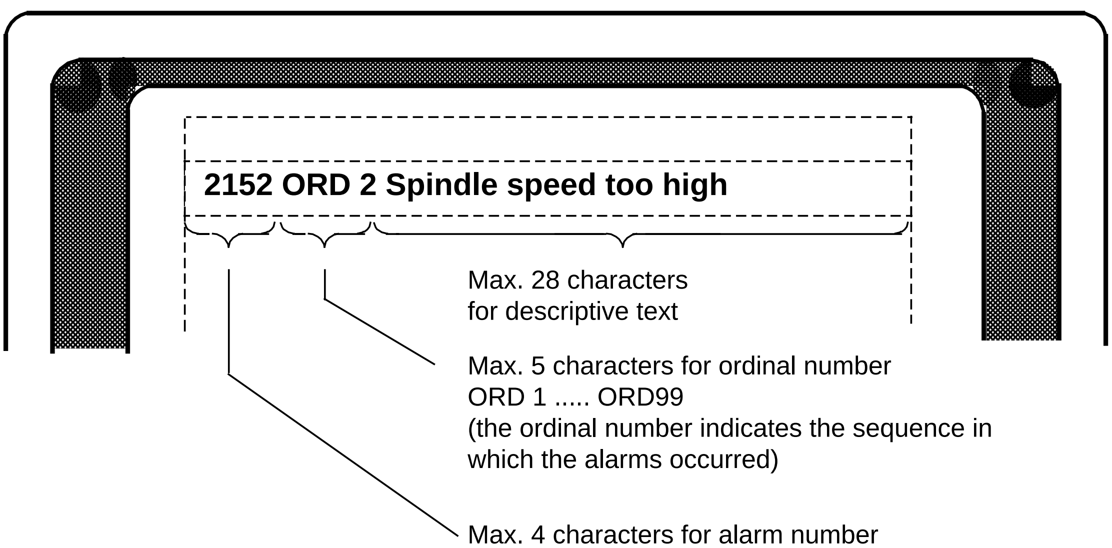


11.5 Display format

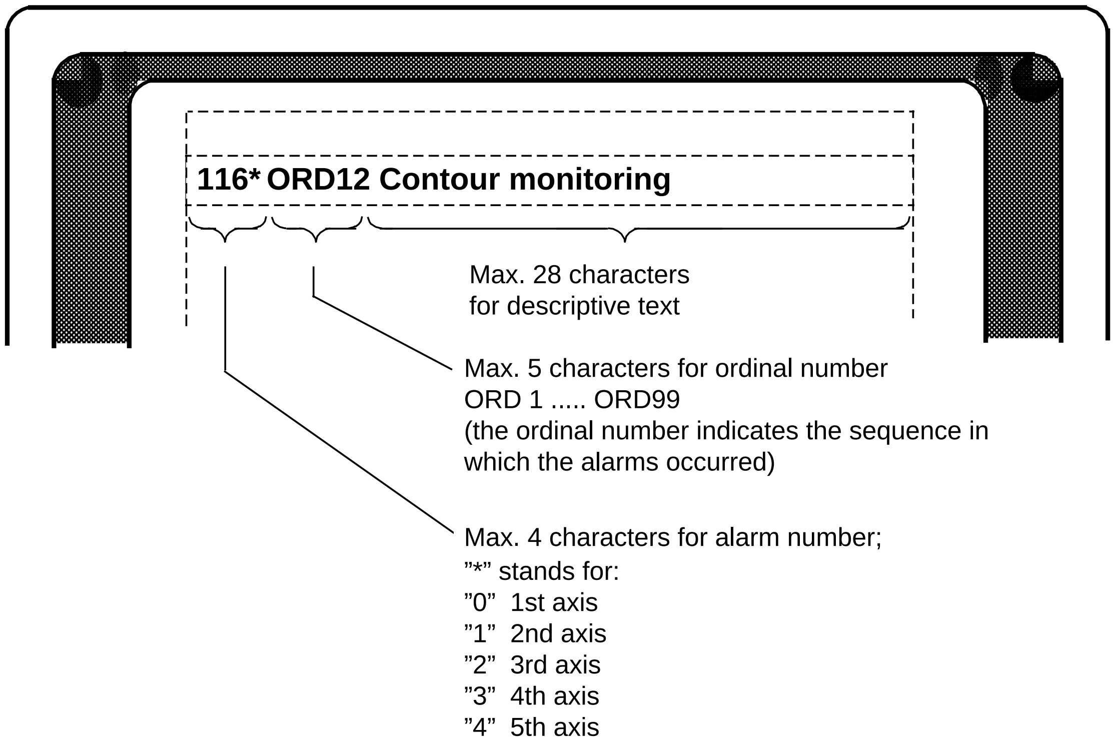
There are 4 types of display format:

- **Example illustrating display format type A**

  Applicable to alarm numbers 0 .... 39 and 2000 2999

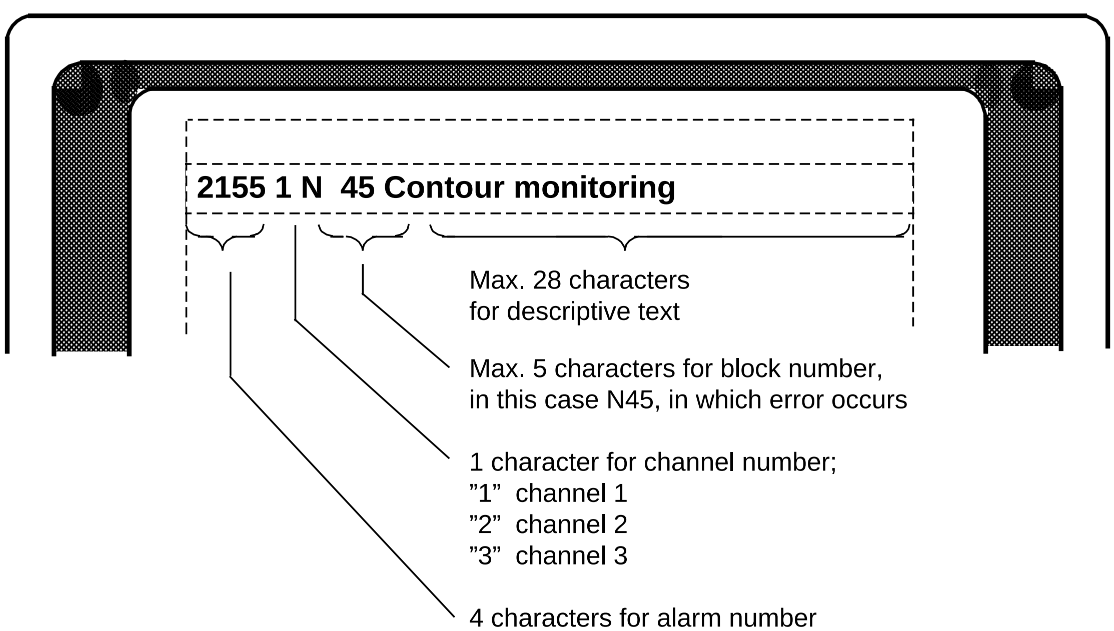


- **Example illustrating display format type B**
  Applies to alarm numbers 1000 1963



- **Example illustrating display format type C**

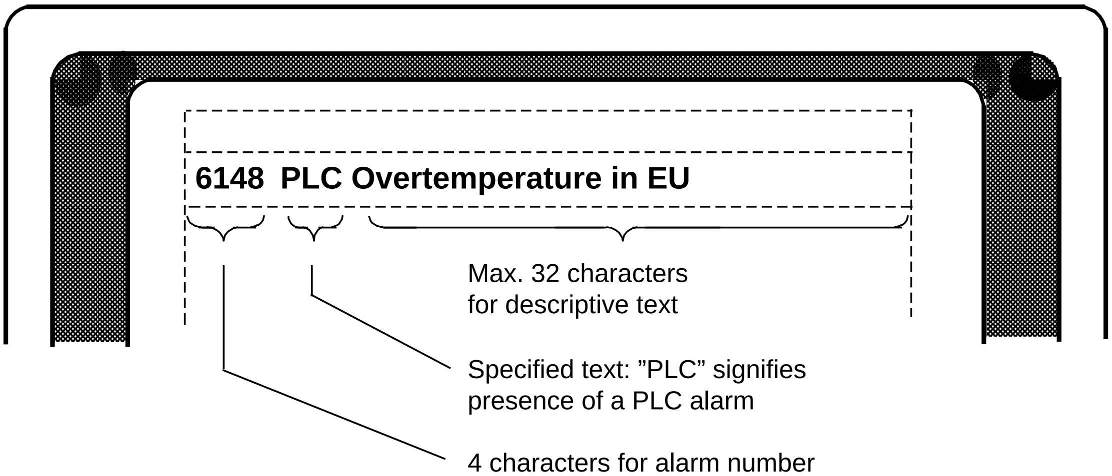
  Applicable to alarm numbers 2000 .... 2999 (partly) and 3000 .... 3055 (partly)



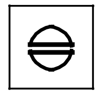
- **Example illustrating display format type D**

  Applicable to alarm numbers 6000 .... 6163 (PLC error messages)

  and to alarm numbers 7000 .... 7063 (PLC operational messages)



11.6 Alarm list POWER ON

1	**Battery alarm - power supply**
Cause: Scan: Explanation: Remedy: Note:	Voltage of back-up battery • With POWER ON • Cyclically Replace battery (see Instruction Manual) The battery voltage has dropped to such a level that buffering of the user memories is only guaranteed for a short period. Replace battery (see Instruction Manual). Acknowledge alarm with  (from power supply unit version 6EV3055-0BC revision ”F”) **Do not switch control off**, otherwise data may be lost ! Dispose of all batteries as special waste.

3	**PLC stop**
Cause: Scan: Effects: Explanation: Remedy:	PLC not ready Cyclically NC START disabled • Setpoint 0 output • NC Ready 2 (NCBB2) cancelled • Servo enable cancelled after time in MD 156 has expired • NC/PLC interface rendered inactive • Resetting of all PLC outputs on I/O board Cyclical and interrupt-driven operation of the PLC is interrupted. Travel with the machine is not possible. Using the PG (programmer) read out the cause of interrupt (ISTACK). Evaluate PLC alarm on NC screen.

4	**Incorrect unit system**
Scan: Effect: Explanation: Remedy:	• With POWER ON • After modifying MD • Conversion factor assumed to be 1 • An illegal combination has been selected (conversion factor greater than ten) for the units of the measuring system (position control resolution) and the unit of the input system. • Position control resolution for rotary axis < $0.5 \cdot 10^{-3}$ degrees and the ordering data option for this is missing. Correct the MD bits then switch the control off and on or include the ordering data option ”Position control resolution for rotary axis < 10^{-3} degrees”

5	**Too many inp. buffer param.**
Scan: Explanation: Remedy:	When formatting user program memory with ”FORMAT USER M. ” softkey. The input buffer parameters (MIB parameters) require so much space in the user memory that the user program memory is now less than 13 Kbytes. Correct MD 5 (enter lower value) and reformat user program memory.

7	**EPROM error**
Scan: Effect: Explanation: Remedy:	• Cyclically • NC START disabled • Output of setpoint 0 • NC Ready 2 (NCBB2) cancelled • Servo enable cancelled after time in MD 156 has elapsed (servo enable relays drop out) • An error was detected while verifying the check sums. • After switching the control off and on, the screen displays which EPROM submodule is defective (see also Section 4.2.2)

8	**Wrong axis/spindle assign.**
Scan: Effect: Explanation: Remedy:	• After modifying MD • NC START disabled • Output of setpoint 0 • NC Ready 2 (NCBB2) cancelled Illegal assignment entered in MD 200* or MD 400* or MD 461*. (e. g. measuring circuit has been assigned to a fictitious axis) (See Section 8).

9	**Memory too small for UMS**
Scan: Effect: Explanation:	With POWER ON UMS declared invalid On switching on, the contents of the UMS are checked and an address list then prepared. This address list requires a certain amount of storage space in the RAM area. UMS lists are too large.

10	**UMS error**
Scan: Effect: Explanation: Remedy:	With POWER ON UMS declared invalid • MD 5015 bit 6 = 1 but no UMS inserted • UMS (RAM) not loaded, i.e. empty • Insert UMS • Load UMS (RAM)

11	**Wrong UMS identifier**
Scan: Effect: Explanation: Remedy:	With POWER ON UMS declared invalid • UMS (RAM) not loaded, i.e. empty • Contents of UMS not defined a) UMS (RAM) overturned b) UMS (EPROM) empty • Wrong UMS inserted • Error when linking with WS 800 • Insert correct UMS • Reload UMS (RAM)

12	**PP memory wrongly formatted**
Scan: Effect: Remedy:	With POWER ON • Error in RAM area of CPU or memory module • The minimum size of the part program was not reached • module withdrawn • Check MD 12 and depress ”CLEAR PARTPR.” softkey.

13	**RAM error on CPU**
Scan: Effect: Remedy:	• With POWER ON • Error in RAM area of module • Format user memory and clear part program in installation overall reset mode • Replace module

14	**RAM error on memory module**
Scan: Effect: Remedy:	• With POWER ON • Error in RAM area of module • Format user memory and clear part program in installation overall reset mode • Replace module

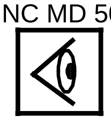
11.7 RS232C (V.24) Alarms

16	Parity error RS232C (V.24)
Effect:	• RS232C (V.24) transmission interrupted • Last block declared invalid
Explanation:	The alarm can only be activated when the setting data ”with parity bit” is set. The parity of the started character (8 data and 1 parity) is incorrect. The alarm has no connection with the RS232C (V.24) character parity error
Remedy:	for ISO or EIA tape (Alarm 23) • Check SD bits 5011, 5013, 5019, 5021 • Test external device

17	Overflow error RS232C (V.24)
Effect:	• RS232C (V.24) transmission interrupted • Last block declared invalid
Explanation:	The external device has transmitted a new character although the NC has not yet processed the old character.
Remedy:	• Check SD bits 5011, 5013, 5019, 5021 • Test external device • Line-controlled or character-controlled transmission • Reduce baudrate

18	Frame error RS232C (V.24)
Effect:	• RS232C (V.24) transmission interrupted • Last block declared invalid
Explanation:	• The number of stop bits is incorrect • Wrong baud rate • Number of data bits is wrong
Remedy:	• Check SD bits 5011, 5013, 5019, 5021 • Test external device • Number of data bits: 7 data + 1 parity

19	**I/O device not ready RS232C (V.24)**
Effect: Explanation: Remedy:	• No files are read in • Low-level DSR signal from external device • Activate external device • Do not use DSR

20	**PLC-alarm-mem. not formated**
Effect: Explanation: Remedy:	• No PLC alarm texts read in The memory for PLC alarm texts was not set up properly (formatted). Sequence: a. Set NC MD 5012 bit 7 b. Key  + NC ON c. ”NC DATA” softkey d. ”FORMAT AL-TEXT” softkey e. RECALL key f. Read in data once more in installation overall reset mode

22	**Time monitoring RS232C (V.24)**
Explanation: Remedy:	• The NC cannot output any characters for 60 seconds – External device blocks CTS (clear to send) signal for more than 60 s – When control signals (DC1- DC4) are used, the external device does not transmit DC1 within 60 s • The NC has not received any characters for 60 seconds • Check external device and switch on • Check and insert cable • Set SD 5017/5025 bit 0

23	**Char. parity error RS232C (V.24)**
Cause: Effect: Explanation: Remedy:	Tape dirty or damaged • RS232C (V.24) transmission interrupted • Last block declared invalid Depending on the definition of program start ”%” or ”EOB”, the NC automatically specifies ISO or EIA code and thus character parity after this character has been received. When the subsequent characters were checked, it was found that one character did ***not*** have the specified parity. Check tape

24	**Invalid EIA char. RS232C (V.24)**
Effect: Explanation: Remedy:	• Data transmission interrupted • Last block declared invalid An EIA character with correct parity has been read in but the character is **not** defined in EIA code. Check tape: Setting data 5026 (EIA code for ”@”), Setting data 5027 (EIA code for ”:”) and Setting data 5029 (EIA code for ”=”) should be checked.

26	**Block > 120 characters RS232C (V.24)**
Cause: Effects: Remedy:	The entered part program block has more than 120 characters. Only characters actually stored are counted (no blanks, no CR, . . .). • Data transmission interrupted • Last block not stored Split block into two or more blocks

27	**Data input disabled RS232C (V.24)**
Cause:	One of the following was read in in normal mode: • NC/PLC-MD without active password • PLC program (PCP), PLC alarm texts (only possible in overall reset mode) In overall reset mode PLC alarm texts were read in and NC MD 5012.7=0.
Effect:	No data stored.
Remedy:	Correct the conditions.

28	**Circ. buffer overflow RS232C (V.24)**
Effect:	• RS232C (V.24) transmission interrupted • Last blocks declared invalid
Explanation:	The signalling rate is so high that the number of characters read in exceeds the number that can be processed by the NC. When the program is re-transmitted, the defective program must first be erased. • RTS signal has no effect at input device (RTS initiates input device STOP) • Signalling rate (baud rate) too high

29	**Block> 254 char. RS232C (V.24)**
Cause:	The block read in has more than 254 characters. All input characters (e.g. blanks) are also included.
Effect:	• RS232C (V.24) transmission interrupted • Last block not stored
Remedy:	Split block into two or more blocks.

30	**PP memory overflow RS232C (V.24)**
Cause:	The maximum storage area for the part program is occupied.
Effects:	• Data transmission interrupted • Last block not stored
Remedy:	Erase old programs and reorganize memory.

31	No free PP number RS232C (V.24)
Cause:	The maximum number of programs specified by means of machine data has been reached.
Remedy:	• Erase old programs and reorganize memory • Modify MD 8 and reformat part program memory Sequence: a. ”SET UP OVERALL RESET” mode b. ”FORMAT USER MEM.” softkey c. ”CLEAR PARTPR.” softkey Old programs are then deleted!!

32	Data format error RS232C (V.24)
Cause:	• Permissible number of decades after an address is incorrect • Decimal point in wrong position • Part programs or subroutines are not correctly defined or terminated (observe pre-header!) • NC expects a ”=” character, but this character is not defined in EIA code
Effect:	Data transmission interrupted Last block not stored
Remedy:	Check data to be read in

33	Different programs same number RS232C (V.24)
Cause:	Read-in and stored programs not identical for same program number Wrong program number in BTR mode.
Effect:	No data stored
Remedy:	Delete old program or rename old program with ”RENAME”
Explanation:	If an existing program with the same program number is read in again, the two programs are compared. If they differ, Alarm 33 is activated. If in BTR mode a program number 0 is preselected in the NC and a program with another program number is offered from outside via the serial interface, then Alarm 33 is given.

34	**Operator error RS232C (V.24)**
Cause: Effect: Remedy:	Data transmission initiated at the NC and the PLC issues a second start signal No data read in Stop data input and restart

35	**Reader error RS232C (V.24)**
Cause: Scan: Effect: Remedy:	Error message from Siemens tape reader • Only if the setting data for the Siemens reader have been set Data transmission interrupted Last block not stored • Restart data transmission • Replace Siemens reader if error recurs

36	**BTR aborted by computer**
Cause: Remedy:	Message frame with error detection received from BTR device Restart

POWER ON alarms

40	**Wrong data in MD 576***
Cause: Effect: Remedy:	Wrong data in MD 576* bit 7, 6, 5 NC START disabled Activate option or check MD 576*

41	**Error absolute submodule**
Cause: Explanation:	Error when using a SIPOS absolute submodule The type of error can be seen from the line ”Status absolute submodule” in the ”Service data axis” display. The error number is displayed. The significance of the error number can be seen from the documentation on the SIPOS absolute encoder submodule.

48	**PLC alarm texts from UMS not allowed**
Scan: Effect: Explanation: Remedy:	• With POWER ON UMS declared invalid If the PLC alarm texts are not read in from tape (NC MD 5012 bit 7) but instead are configured using the WS800 programming workstation, the texts are transferred to the UMS together with cycles, displays and menus. On checking the inventory of the UMS, the control has established the presence of both PLC alarm texts from tape and alarm texts in the UMS. • Reset NC MD 5012 bit 7 • Check UMS and re-configure if necessary using WS800

87	**Illegal software limit switch**
Cause: Effect: Comment:	An illegal value has been entered in the machine data for software limit switch MD 224*, 228*, 232*, 236*) or prelimit switch (MD 376*). • Removal of NC-READY2 • Locking of NC START The check is performed effectively after POWER ON, independently of NC MD 560* bit 5 ”Working area limitation, Software limit switch”.

11.8 Axis-specific RESET alarms

104*	**DAC limit reached**
Scan: Effect: Explanation: Remedy:	Cyclically No direct effect. The error enters the following error Alarm 156* Setpoint at DAC is higher than input in MD 268* (max. DAC setpoint). No further increase in setpoint possible! • Operate at lower speed • Check actual values (pulse generator) • Check MD 268* • Check drive actuator (tachogenerator compensation at Vmax.) • Check MD 364* and MD 368* (var. increment weighting)

108*	**Actual value difference overflow**
Scan: Effect: Explanation: Remedy:	With each axis movement (including in follow-up mode) • NC START disabled • Setpoint 0 • NC Ready 2 cancelled • Servo enable cancelled after time in MD 156 has expired • Follow-up mode • Actual machine value is lost (wrong position) • With high-speed axis travel the register has overflown in the event of an error. The reference point was lost in the process. • Reduce maximum speed. (Depending on MD 364* and 368*) • Check MD for variable increment weighting (MD 364* and MD 368*).

112*	Clamping monitoring
Cause:	• Incorrect position feedback polarity • Mechanically clamped axis forced out of position • Fault at control device (actuator), tacho-generator, motor, mechanical components or NC measuring-circuit hardware
Scan:	• At rest • During clamping • During deceleration
Effect:	• NC START disabled • Setpoint 0 • Servo enable cancelled after time in MD 156 has expired • Follow-up mode
Explanation:	• During positioning the following error could not be reduced more rapidly than the time input in MD 156 • During clamping the limit specified in MD 212* was exceeded
Remedy:	• MD212* (clamping tolerance) must be greater than MD 204* (exact stop limit coarse). • MD156 (servo enable cutoff delay) must be such that the following error can be suppressed within this time

116*	Contour monitoring
Scan:	• During processing in automatic mode but not: during acceleration during deceleration at speeds less than in MD 336* (contour speed)
Effect:	• NC START disabled • Setpoint 0 • Servo enable cancelled after time in MD 156 has expired • Follow-up mode
Explanation:	• Tolerance band MD 332* exceeded at a speed greater than in MD 336* • During acceleration or braking the axis has not reached the new speed within the time specified by the servo gain factor
Remedy:	• Increase tolerance band MD 332* • Check servo gain factor • Check optimization of speed controller • Check drive actuator

132*	**Control loop hardware (POWER ON required)**
Scan: Effect: Explanation: Remedy:	• Cyclically • NC START disabled • Setpoint relay drops out • NC Ready 2 cancelled • Servo enable cancelled after time in MD 156 has expired • Follow-up mode Measuring-circuit differential signals • are not in phase • are short-circuited to frame • are missing altogether • Check whether measuring-circuit connector has been inserted • By connecting measuring-circuit short-circuit connector, it is possible to check whether the measuring-circuit module is in order • Check differential signals using oscilloscope • Replace encoders (See Section 7 for characteristics of measuring-circuit differential signals)

136*	**Meas. system dirty**
Scan: Effect: Explanation: Remedy:	Cyclically NC START disabled. The active program is fully executed. In measuring systems with contamination signal (e.g. EXE) an error is signalled by the measuring system to the NC. Check measuring system

148*	**+ Software limit switch**
152*	**- Software limit switch**
Scan: Effect: Explanation: Remedy:	With each axis movement • NC START disabled • Setpoint 0 Alarm is only active after reference point approach • Software limit switch 1 or 2 approached, depending on PLC interface signal ”2nd software limit switch active” • Move away from limit switch in reverse direction • Check MD 224*, 228*, 232*, 236*

156*	**Set speed too high**
Scan: Effect: Explanation: Remedy:	• Cyclically • NC START disabled • Setpoint 0 • Servo enable cancelled after time in MD 156 has expired • Follow-up mode Set speed output in control is higher than specified in MD 264* • Motor could not follow set speed input • Check whether value in MD 264* is greater than in MD 268* Check drive • Check measuring system • Earthing neutral point at NC ? • Check drive actuator • Check direction of position control (setpoint/actual value inverted?)

160*	**Drift too high**
Scan: Effect: Explanation: Remedy:	• Cyclically • NC START disabled • Green LED ”Position not yet reached” lights up • No traversing movement possible The drift to be compensated by the NC has exceeded approx. 500 mV. • Perform drift compensation in MD 272* (see also Section 8) Operator input: – Select MD 272* – Depress EDIT key • Check whether drift was adjusted correctly at drive unit • Check drive actuator • Check earthing

168*	**Servo enable trav. axis**
Scan: Effect: Explanation: Remedy:	With each axis movement • NC START disabled • Setpoint 0 • Servo enable cancelled after time in MD 156 has expired • Follow-up mode Axis-specific servo enable cancelled by PLC user program during traversing movement • Check PLC program

172*	**+ Working area limit**
176*	**- Working area limit**
Scan: Effect: Explanation: Remedy: Note:	During processing in automatic mode During axis traversing in JOG, INC and REPOS mode • NC START disabled • Setpoint 0 Working area limitation in the setting data has been reached. • Check working limitation in setting data (select using ”Setting Data”and ”Common Axial” soft keys) • Check program • Working area limitation in JOG mode only active when NC MD 5003 bit 6 is set

180*	**Axis in several channels**
Cause: Effect: Remedy:	During simultaneous processing of two programs in different channels, one axis has been programmed in both programs (channels) such that a traversing movement for the axis in question is output by both programs. • NC START disabled • Setpoint 0 Check both programs

184*	**Stop behind reference point**
Scan: Effect: Explanation: Remedy:	During approach to reference point • NC START disabled • Setpoint 0 • Reference point not reached During approach to the reference point the axis was stopped between the reference cam and zero mark of the measuring system. Repeat approach to reference point

196*	**Follow-up/park for axis**
Explanation:	• The interface signal PARK for the axis is active • Changeover from spindle mode to rotary axis mode has not yet taken place

11.9 General RESET alarms

2000	**Emergency stop**
Scan: Effect: Significance: Remedy: Caution:	Cyclically • NC START disabled • Setpoint 0 • Servo enable cancelled after time in MD 156 has expired • Follow-up mode PLC outputs ”EMERGENCY STOP” signal to NC. • Check with PLC-STATUS whether Q 78.1 = 0 • Check whether EMERGENCY STOP cam approached or EMERGENCY STOP pushbutton actuated • Check PLC program According to legal requirements the EMERGENCY STOP status must be selected not only by the control (software) but also by the hardware (using relays).

2030	**Wrong path increment**
Cause:	• Set part position too large with spline. Incorrect spline coefficient K, path line I or axis position • Axis offset not carried out before spline selection

2031	**Eval. factor too high/low MD 388***
Scan: Effect: Remedy:	On every axis movement • NC START disabled • Setpoint 0 • Stop in processing Check MD 388*

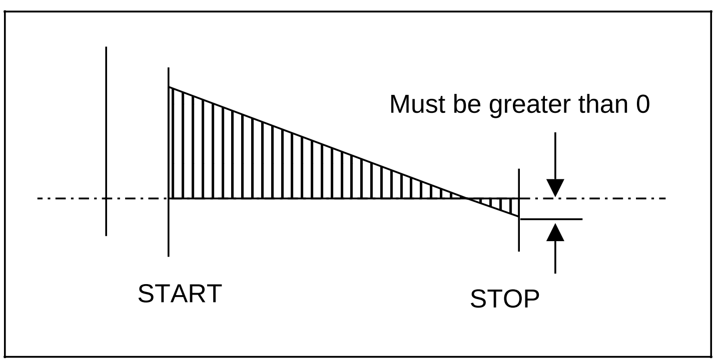
2032	**Stop during threading**
Effect: Explanation:	• Setpoint 0 • NC Start disabled • Stoppage has occurred in feedrate per revolution during thread cutting, resulting in destruction of the thread. • Axis-specific feed enable was cancelled during ”G33 active”

2034	**Speed reduction area**
Explanation:	The software pre-limit switch has been overrun and the axes braked to reduction speed.
Remedy:	• Check program • MD0 - Pre-limit switch • MD1 - Speed behind pre-limit switch (reduction speed)

2035	**Feed limitation**
Cause:	An axis speed was specified which was larger than the maximum axis speed (MD 280*)
Effect:	Speed is reduced to maximum speed

Explanation:	• The programmed speed is greater than the tool path feedrate resulting from the maximum speeds of the axes.
	• During thread cutting G33 a spindle speed and a thread pitch were programmed which would lead to an axis speed greater than the maximum axis speed (MD 280*).
	• One of the following data is greater than the maximum speed
	  • conventional feed (JOG)
	  • conventional rapid traverse (JOG)
	  • incremental speed
	  • reference point approach speed

2036	**G35 thread lead decr. error**
Scan:	During thread cutting.
Significance:	The lead decrease in the thread is so great that a diameter of 0 or less would result at the end of the thread.



Remedy:	Program smaller lead decrease or shorter thread.

2037	**Prog. S value too high**
Explanation:	Programmed, overstored spindle speed ”S” exceeds ”16 000”. Enter slower spindle speed (S value limited in control to ”16 000”).

2038	**Path feed too great**
Explanation: Effect: Remedy:	• The value entered in the machine data ”Max. velocity” exceeds the permitted value range. • The programmed path of velocity results in an axial velocity which is outside of the value range. The axis number is displayed after N. Blocking of NC Start and dropping out of setpoint relay. Check the MD ”Max. velocity”. Observe the value range.

2039	**Reference point not reached**
Scan: Effect: Significance: Remedy:	• In AUTOMATIC/MDI mode after NC START • NC START disabled The reference point was not approached in all defined axes • Approach reference point in axes concerned. • Set NC MD 5004 bit 3 (NC START without reference point). **Caution:** No software limit switches active. • Set NC MD 560* bit 4. Approach to reference point can then be suppressed for one or more special axes on an axis-specific basis. **Caution:** No software limit switches active in these axes.

2040	**Block not in memory**
Effect: Explanation:	• NC START disabled • The block number sought in block search is not available in the program. • With a jump in the program, the programmed block number could not be found in the specified direction.

2041	**Program not in memory**
Effect: Explanation: Remedy:	• NC START disabled • Preselected program is not available in memory • Subroutine called in program is not available in memory • Preselect correct program, check program overview

2042	Parity error in memory
Scan: Explanation: Effect: Remedy:	• During processing in automatic mode One or more characters are deleted in the memory, so they can longer be recognized (these characters are output as ”?”) NC START disabled • Correct program in EDITOR or, if necessary, delete complete block and re-input • With a large number of ”?” the complete memory may have been erased; in this case check the battery and reformat the part program memory

2043	Progr. error on transform.
Explanatio:n	• Programming of real axes of the transformation group when transformation has been selected. • Programming of fictitious axes when transformation has been deselected. • Programming traversing movements in the selection block

2046	Block > 120 characters
Scan Effect: Explanation: Remedy:	• During processing in automatic mode • NC START disabled ”LF” is inverted in the memory producing a block of more than 120 characters. Insert ”LF” or delete complete block

2047	Option not available
Effect: Explanation: Remedy:	• NC START disabled The programmed function is not included in the control's function set. Correct program, check MD

2048	**Circle end point error**
Effect:	• NC START disabled • Programmed circle end point not on circle • End point is further away by more than the input limit in MD • No geometry in first block of contour subroutine with L95 stock removal cycle
Remedy:	Correct program

2056	**Travel thru transf. center**
Explanation:	• Radius axis (linear axis) tends to travel to the zero point of fictitious coordinate system on the programmed path
Remedy:	• Change cutter radius • Program different path motion

2057	**Opt. thread/rev. not available**
Significance:	• A thread has been programmed with G33, G34, G35 although this function is **not** implemented in the control • Feedrate per revolution G95 has been programmed
Remedy:	• Correct program • Check MD

2058	**3D option not available**
Explanation:	• 3 axes programmed simultaneously • Programmed block results in movement of 3 axes
Remedy:	• Correct program, check MD

2059	**G92 Program error**
Explanation:	• Use of an illegal address character G92 is only allowed with address ”S” (programmed spindle speed limitation for G96) or ”P” (cylindrical interpolation).

2060	**TO, ZO Program error**
Significance:	• Unavailable tool offset number selected • Selected zero offset or tool offset value too large • Type (P1) of tool offset called defined with 0

2061	**General program error**
Effect: Remedy:	NC START disabled • Contour calculation not possible • Incorrect machine data for axis duplication function

2062	**Feed missing/not prog.**
Cause:	• No F value programmed • F value too small (machine data) • Programmed feedrate per revolution G95 greater than 50 mm/rev • No feedrate per revolution programmed • With G98, traversing path=0 for the rotary axis or the feedrate value has been omitted.
Remedy:	Program feed correctly

2063	**Thread lead too high**
Effect:	NC START disabled
Explanation:	Thread pitch of more than 400 mm/revolution (16 inches per revolution) programmed.
Remedy:	• Program smaller thread pitch • Possibly run program on a machine with SINUMERIK 850 (max. speed 2000 mm/revolution)

2064	**Program error round axis**
Scan: Explanation: Effect: Remedy: Note:	During processing in automatic/MDI mode In the case of rounding to half or full degrees with a rotary axis, the control monitors the programmed positions to ensure that rounding has been correctly performed. • NC START disabled • Programmed travel in block is **not** processed • Program correct position in rotary axis • Check MD 560* bits 2 and 3 In manual modes JOG, INC the control rounds automatically to valid values; in AUTOMATIC or MDI mode it only monitors the programmed positions and does not round automatically.

2065	**Pos. behind SW overtravel**
Scan: Effect: Explanation: Remedy:	During processing in automatic/MDI mode • NC START disabled • Programmed travel is **not** processed Programmed end position of block is behind software limit switch. • Check program, tool offset and zero offset • Check MD224*, 228*, 232*, 236* as a function of PLC interface signal ”2nd software limit switch active”

2066	**Thread lead incr./decr.**
Scan: Effect: Explanation: Remedy:	During processing in AUTOMATIC/MDI mode NC START disabled Thread or pitch increase or decrease of more than 16 mm per revolution (0.6 inch/revolution) programmed. Program smaller thread lead increase/decrease

2067	**Max. speed = 0**
Scan: Effect: Explanation: Remedy:	During processing in AUTOMATIC/MDI mode NC START disabled The maximum speed of the axis programmed in the block is ZERO. Check MD 280*

2068	**Pos. behind working area**
Scan:	During processing in AUTOMATIC/MDI
Effect:	• NC START disabled • Programmed travel **not** processed
Explanation:	Programmed end position of block is behind working area limitation in one or more axes.
Remedy:	• Check working area limitation (positive and negative) • Modify working area limitation by means of G25/G26 in program

2072	**Incorrect input value (contour definition)**
Explanation:	Value input not calculable for contour definition calculation.

2073	**No intersection point (contour definition)**
Explanation:	No intersection is obtained with programmed values when calculating contour definition.

2074	**Incorrect angle value (contour definition)**
Explanation:	• Angle 360° programmed • Angle value not practical for defined contour

2075	**Incorrect radius value (contour definition)**
Explanation:	• Radius too large • Radius not permitted with defined contour

2076	**Incorrect G02/G03 (contour definition)**
Explanation:	Circle direction **not** possible with defined contour

2077	**Incorrect block sequence (contour definition)**
Explanation:	Several blocks are required for calculating contour definition: • Block sequence incorrect • Data not sufficient (under-determined) **Example:** N10...B15 LF N20...G3 I20 LF

2078	**Incorrect input parameter (contour definition)**
Explanation:	• Programmed parameter sequence not allowed • Parameter sequence incomplete for defined contour **Example:** N10...X60 B15 LF (Z axis missing) N20...X90 B10 LF

2081	**CRC/TNRC not allowed**
Explanation:	With cutter/tool nose radius compensation (CRC/TNRC, G41/G42) selected, the following functions must not be programmed: G33, G34, G35, G58,G59, G92, M19 S...,
Remedy:	• Cancel CRC/TNRC with G40 or program D0

2082	**CRC plane not determinable**
Significance:	Axes of selected CRC plane do not exist.
Remedy:	• Check MD 548*, 550*, 552* • Select correct plane with G16

2087	Coord. rotation n. allowed
Explanation:	When coordinate rotation has been programmed in the NC program a circular motion is to be performed immediately after changing the overall rotating angle.
Remedy:	Check NC program

2088	Battery alarm abs. submod. 1
Cause:	Battery voltage too low Battery test every 10 minutes
Remedy:	Replace battery submodule on absolute submodule 1 with control under power.

2089	Battery alarm abs. submod. 2
Cause:	Battery voltage too low Battery test every 10 minutes
Remedy:	Replace battery submodule on absolute submodule 2 with control under power.

2152	Spindle speed too high
Scan: Explanation:	• Only when MD 5200 bit 2 is set (pulse generator available) • The spindle actual speed has exceeded the tolerance defined by the machine data.
Effect:	• Setpoint for all axes set to ”0” • Servo enable of the axes is cancelled when the delay in MD 156 has elapsed. • Follow-up mode for all axes is given (”Axis in position control” is removed) • Spindle is halted (setpoint 0)
Remedy:	• Program smaller S value • Check MD 403*-410* (max. spindle speed for first to eighth gear speed) • MD 445* (tolerance band of the max. spindle speed) • MD 451* (max. spindle speed) • Gear speed correctly selected by the PLC? • G 92 S... incorrectly programmed for v constant (G96)

2153	Control loop spindle - HW
Scan: Effect: Explanation: Remedy: Note:	Cyclically • NC START disabled Setpoint 0 • NC Ready 2 cancelled • Spindle servo enable cancelled after time in MD 4470 has expired As for Alarm 132* As for Alarm 132* Axis measuring-circuit alarms which occur after the alarm in the order control loop - spindle - hardware are recognized but not displayed.

2154	Spindle meas. system dirty
Scan: Explanation: Effect: Remedy:	Cyclically In measuring systems with a contamination signal, the measuring system has signalled an error to the NC. NC START disabled Check measuring system

2155	Option M19 not available
Scan: Effect: Explanation: Remedy:	• During processing in AUTOMATIC/MDI mode NC START disabled "M19 S..." programmed in part program although this function is not available. • Correct program • Retrofit option "M19"

2160	Scale factor not allowed
Effect: Significance: Remedy:	Machining halted Range of values from 0. 000 01 to 99, 999 99 exceeded Check NC program block with G51 P..

2161	Scale change not allowed
Effect: Significance: Remedy:	Machining halted Scale change not allowed in NC program Check NC program with G51 X.. Y.. Z.. U.. P

2171	Approach not possible
Explanation:	The control supplements no more than one axis in the programmed plane. Approach is not possible when two axes in the programmed plane are to be supplemented.
Remedy:	• Check NC program for complete axis programming in approach block. • Programming of cancellation block immediately after selection block not allowed (no tangent can be calculated).

2172	Retract not possible
Explanation:	See Alarm 2171
Remedy:	• Check NC program for complete axis programming in approach block. • An approach movement must be programmed with G48 progr. cancel movement (retract as for approach).

2173	Wrong appr./retract plane
Explanation:	The select/cancel movements for the smooth approach/retract function are plane-related to the selected plane G16, G17, G18, G19.
Remedy:	Check NC program as to whether a change of plane is programmed in the block after selection or in the cancellation block.

2183	N564* AXIS IS NOT A ROTARY AXIS
Explanation:	Changeover to rotary axis although this axis has not been declared as rotary axis. The relevant machine data is specified after N.

2184	M function for C axis invalid
Explanation:	M functions reserved by the NC have been used for selecting/deselecting the rotary axis mode (e.g. M01)

2189	Transformation undefined
Explanation:	• Transformation type not defined • Option bit not set • Transformation axes defined several times or incorrectly

2190	Transformation axes assigned
Explanation:	For transformation selection one of the real axes of the transformation grouping is programmed in another channel.

2191	Transformation in zero
Explanation:	1st real axis located in zero for transformation selection
Remedy:	Remove axis from zero in JOG, INC mode

2192	Axis doubling active
Cause:	PLC signals QB 81.2 or 81.3 changed outside ”Reset state”. Travel key activated in ”jog mode” without Automatic interrupted and QB81.2 and 81.3 and MD 5019.0 set. Axis duplication active and axial PLC signals (Servo enable, Feedrate enable, Mirror, Follow-up mode, Axis disable) not identical for both axes. PLC signals QB 81.2 and 81.3 both equal to zero, but option bit set.
Effect:	NC start disabled
Remedy:	Set PLC signals correctly

2193	**No additional axes possible**
Cause:	This occurs only at NC Start after block search in the target block when the MD "Add axis after block search" have been set: • axes are to be added in G36 blocks • axes are to be added in G98 blocks without axis traverse motion.
Abhilfe:	Do not perform block search on this block.

2194	**G36 position rotary axis missing**
Cause:	Occurs only if a G68 is to be generated for the rotary axis in a G36 block when MD 572* bit 2=1.
Remedy:	• Program the rotary axis first of all with G0 or G01 (e.g. G0 91 C=0). • Select target block in such a way that no G36 is active in the target block and that the rotary axis is already programmed before the G36 block.

11.10 RESET alarms spindle-specific

225*	Spindle speed too high
Scan: Explanation: Remedy:	Only when NC MD 520* bit 2 is set (encoder available) The actual spindle speed is greater than that set in the machine data or in the setting data. • Program smaller S value • NC MD 403* to 410* (max. spindle speed for 1st to 8th gear stage) • NC-MD 445* (tolerance band of max. spindle speed) • NC-MD 451* (max. spindle speed) • Check gear stage of PLC • Check G92 S ... for ”v=constant” • Check setting data for spindle speed limitation • Program G26 S ...

226*	Control loop spindle hardware (POWER ON ALARM)
Scan: Effect: Explanation: Remedy:	Cyclically • Locking of ”NC START” • Drop-out of setpoint relay, setpoint 0 • Removal of NC Ready2 • Controller enable of the spindle is removed after expiry of time in MD 447* • As for alarm 132* • As for alarm 132*

227*	Contamination measuring system (spindle)
Remedy: Effect: Explanation: Remedy:	Cyclically Locking of ”NC Start” In measuring systems with contamination signal, an error has been signalled from the measuring system to the NC. Check the measuring system.

228*	Option M 19 missing
Scan: Effect: Remedy:	The interface signal ”M 19 from PLC” is on although this function is not implemented in the control. Locking of ”NC START” • Check program • Check NC MD • Retrofit option E42

11.11 Alarm list ACKNOWLEDGE

3000	**General program error**
Explanation:	A general programming error which cannot be precisely defined has been made in a block in the program. Example: • The programmed axis is not available at the machine. • Incorrect interpolation parameters programmed. • Axis duplication option active and D number in part program greater than 49. • Spindle was programmed in rotary axis mode.
Remedy:	Check defective block in "Correction Block" Cursor is set in front of incorrect word if possible. Number of the defective block is behind the alarm no. in the alarm line.

3001	**Geometry parameters >5**
Explanation:	More than 5 geometry parameters such as axes, interpolation parameters, radii, angles etc. have been programmed in the block.
Remedy:	As for Alarm 3000

3002	**Polar/radius error**
Explanation:	No programming in block with polar/radius programming for: • Angle • Radius • Centre point coordinates
Remedy:	As for Alarm 3000

3003	**Invalid address**
Explanation:	• The address programmed is not defined in the machine data. • The axis names for basic setting plane (MD 548*, 550*, 552*) do not correspond to the defined axis designations (MD 568*) • Axis duplication option active and tool system 2 axes programmed in part program.
Remedy:	MD 108*...MD 118* are not correctly assigned The MDs for I, J, K parameters (MD 304*) contain a number not equal to 0, 1, 2 or 3. • As for Alarm 3000 • Correct machine data

3004	CL800 error
Explanation:	• @ function not available • Incorrect address after @ • Number of addresses after @ incorrect • Value in K, R or P not permissible • Number of decades excessive • No decimal point allowed • Jump address incorrectly defined • System memory (NC MD, PLC MD, TO, ...) not available • Bit number excessive • Sine or cosine angle incorrectly stated • MD write @ inhibited MD 5012.2=1
Remedy:	• @ as per Programming Guide • Only addresses allowed are K, R and P • Jump addresses forwards with ”+” backwards with ” - ” • Check validity of values in stated addresses • If necessary select decoding single block (DEC-SBL) and re-check program

3005	Contour definition error
Explanation:	Coordinates in blueprint programming have been defined such that no intersection is produced.
Remedy:	As for Alarm 3000

3006	Wrong block structure
Explanation:	• More than 3 M functions programmed in block • More than 1 S function programmed in block • More than 1 T function programmed in block • More than 1 H function programmed in block • More than 4 auxiliary functions programmed in block • More than 3 axes with G00/G01 programmed in block • More than 2 axes with G02/G03 programmed in block • G04 programmed with addresses other than ”X” or ”F” • M19 programmed with addresses other than ”S” • Incorrect or no interpolation parameter for G02/G03 (MD 304*) • G92P... not alone in block • G74 not alone in block • Spline interpolation not allowed • G98 active without G0, G1 or G36 • Contour elements are traversed with G98 feed • In the G36 block there is no thread lead or the thread lead is not assigned through MD 304* to the infeed axis • In a G98 or G36 block, TRANSMIT/cylindrical interpolation is active with the same rotary axis • Wrong number of rotary axes with G98 • Wrong number of axes with G36 • Radius, angle or L or P have been programmed in a G35 block • Because of the thread lead and the distance to go for the infeed axis, the distance to go in the rotary axis is to great with G36.
Remedy:	As for Alarm 3000

3007	**Wrong setting data program**
Explanation:	• G25 / G26 programmed • G92 programmed with an address other than ”S” or ”P” • M19 programmed with an address other than ”S”
Remedy:	As for Alarm 3000

3008	**Subroutine error**
Explanation:	• M30 programmed as program end • M17 missing at program end • 5th nesting depth activated (only 4 subroutine levels possible with SINUMERIK 810) • M17 programmed in main program
Remedy:	As for Alarm 3000

3009	**Program disabled**
Explanation:	L0 preselected in AUTOMATIC display (is prohibited)

3010	**Intersection error**
Significance:	This error may occur in conjunction with stock removal cycle L95 if: • Contour program programmed without G0, G1, G2, G3 • @ 714 programmed in contour program • Incorrect plane in contour program • No intersection found • More than quarter circle programmed in contour program • No geometry in first block of contour program of L95 stock removal cycle
Remedy:	As for Alarm 3000

3011	Number of axes > 2 / axes twice
Explanation:	• An axis has been programmed twice in the same block. • More axes have been programmed than are available at the machine
Remedy:	As for Alarm 3000

3012	Block not in memory
Explanation:	• Program not terminated with M02 / M30 / M17. • Block number stated in the jump (@ 100, 11x, 12x, 13x) was not found in the specified direction.
Remedy:	As for Alarm 3000

3013	Simulation disabled
Explanation:	With the machine data set accordingly, graphic simulation (for checking the part program) is only possible if no program is being processed simultaneously at the machine (depending on MD 5007, bit 4).
Remedy:	• Interrupt the part program at a suitable point using the RESET key • Process the part program up to the end, then simulate

3016	External data input error
Effect:	Data transfer interrupted.
Explanation:	With external data input from the PLC to the NC: • Code is incorrect • Value excessive • Dimensional identifier impermissible • Option not available
Remedy:	• Check PLC program • Check NC MD, PLC MD

3017	**Part program no. occurs twice**
Scan: Effect: Explanation: Remedy:	With "POWER ON RESET" (switching on of control) NC START disabled The EPROM card for the cycles has a part program in duplicate Check UMS

3018	**Distance from contour too great**
Scan: Effect: Explanation: Remedy:	After NC start (automatic) • Stop in processing The distance from the circle contour (MD 9) is too large after repositioning Note MD 9, move closer to the contour

3019	**Option RS 232 not available**
Explanation: Remedy:	The second RS 232 C (V.24) interface has been activated by the PLC or using the soft key without the option being available. • Transmit data via the first RS 232C interface • Retrofit option C62 (second RS 232C interface)

3020	**Option not available**
Explanation: Remedy:	A function has been programmed which is not available in the control. • As for Alarm 3000 • Retrofit option

3021	**CRC/TNRC contour error**
Scan: Explanation:	• With CRC/TNRC selected **NOT:** • in selection block • in cancellation block Correction calculation produces a traversing movement opposing the programmed movement.

3024	Display description not available
Explanation:	A configured softkey has been used to skip to a display which is not available in the user memory submodule or system memory.
Remedy:	• Check display number • Check softkey function

3025	Display description error
Explanation:	• A display with graphics has been configured but the control does not have the ”Graphics” option. • The selected display has too many variables or fields. • A display type not known to the control has been configured.
Remedy:	• Check display with programming workstation • Retrofit ”Graphics” option if necessary

3026	Graphics / text too volum.
Explanation:	• Configuring error in selected display • Sum of graphics and text elements too large
Remedy:	• Check display with programming workstation • Split contents over 2 displays if necessary

3027	Graphics command too volum.
Explanation:	Sum of graphics commands in selected display too large.
Note:	This alarm activates Alarm 3026.
Remedy:	• As for Alarm 3026

3028	Too many fields / variables
Explanation: Remedy:	• Configuring error in selected display. The number of fields or variables is limited in view of the specific length of the transfer buffer. A maximum number of fields/variables cannot be stated since the fields/variables can have different formats and positions. • Check display with programming workstation • Reduce number of fields/variables • Split contents over 2 displays if necessary

3029	Graphics option not available
Explanation: Remedy:	Graphics elements have been configured in the selected display although NC MD 5015 bit 2 ”Graphics” is not available in the control. • Set bit 2 of NC MD 5015 • Configure displays without graphics elements

3030	Cursor memory not available
Explanation: Remedy: Note:	The cursor memory configured in the selected display is not correct (number not permitted or too large). Re-determine cursor memory with programming workstation. • The function of the cursor memory is to place the cursor at its original position when the display is called once more.

3032	Too many fields / variables (DIS GGS)
Note:	As for Alarm 3028

3033	Display text not available
Explanation: Remedy:	An error has occurred during linking with the programming workstation. Check link list and re-link with programming workstation (watch for link errors!).

3034	Text not available
Explanation:	The followings texts have been incorrectly linked or not linked at all in the selected display: • Menu texts • Dialog texts • Mode texts • Alarm texts etc.
Remedy:	Check display using programming workstation

3040	Fields / var. not displayable
Explanation:	• Field/variable incorrectly configured or not configured • Field/variable configured with inadequate number of positions • Field/variable overflow (range of values exceeded)
Remedy:	Check field/variable using programming workstation; if necessary delete and re-input.
Note:	If the error occurs in the Siemens standard displays, the range of values has been exceeded.

3041	Too many fields / variables (DID DIS)
Explanation:	As for Alarm 3028

3042	Display description error
Explanation:	An error has been determined in the display description which cannot be accurately assigned, e.g. a non-existent field has been configured (NC MD for 5th axis).
Remedy:	Check display using programming workstation, graphics not available

3043	Display description error
Explanation:	As for Alarm 3024
Remedy:	As for Alarm 3042

3046	**Variable error**
Explanation: Remedy:	A variable has been selected which cannot be represented in control. Check display using programming workstation; re-input variable if necessary

3048	**Wrong workpiece definition**
Explanation:	Minimum and maximum values have been reversed when defining the workpiece. **Example:** Xmin. = 100 Xmax. = 50
Remedy:	Check workpiece definition for valid values.

3049	**Wrong simulation area**
Explanation:	No values or incorrect values have been input when defining the simulation area.
Remedy:	• Check values for simulation area (all levels). • Simulation may only be restarted after depressing RESET and the ACKNOWLEDGE ALARM key.

3050	**Incorrect input**
Explanation:	Simulation data incorrect/not defined.

3063	**Data block not available**
Explanation: Remedy:	In PLC STATUS a DB No. was selected, which is not available. Select or create correct DB.

3081	CRC not selected on approach
Explanation:	The function ”soft contour approach and retract” is only possible when cutter radius compensation has been selected. G41/ G42 D0 is then considered to have been selected.
Remedy:	Select CRC

3082	Feed missing/not prog.
Explanation:	When simulating a part program: • No feedrate F programmed • F value too small (MD) • Wrong feed type in G36 block
Remedy:	Programm feedrate correctly

3083	Feed limit fictitious axis
Explanation:	The feedrate is reduced block by block if necessary to ensure that the rotary axis is not moved faster than specified in MD 280* when the Transmit function is active. Alarm ist not actuated when rapid reverse ist programmed in the part program (G00, G10)
Remedy:	Programm smaller feedrate or select other cutter radius

3084	Wrong data in DB 39
Explanation:	• R parameter no. not within permissible range • Incorrect assingment axis /cam parameter
Remedy:	Correct DB 39 values and specify value transfer

3087	Transformationsdaten-Fehler
Erläuterung:	Fehlerhafter Inhalt in Transformationsmaschinendatum (wird nur bei Neustart gemeldet). Der Alarm ist satzbezogen. Die Nummer des fehlerhaften Maschinendatums wird an Stelle der Satznummer angegeben.

3200	e.g. 1N5 illegal working area limitation
Explanation:	N5 = axis number 5 (a number from 1 to 7 is possible) 1 =min. working area limitation 2=max. working area limitation This alarm is issued when the entry for working area limitation is beyond the permitted values. Together with this alarm, the values are limited to the maximum possible. This alarm sinply indicates that this limitation has taken place.

3201	Spindle not synchronized
Explanation:	Initiation of C axis mode without the spindle being synchronized. This can occur if the spindle has not been rotated after POWER ON.
Remedy:	Rotate spindle.

3202	M19 still active
Explanation:	Initiation of rotary axis mode although M19 is still active.
Remedy:	Terminate M19.

6000 : 6063	**PLC user alarm**
Explanation: Remedy:	Initiation bit was set in the PLC user program Check PLC program or machine function

6100	**Signal converter missing**
Cause: Effect: Remedy:	Load or transfer command to unavailable peripheral device (I/Os), e.g. L PB, T PB PLC STOP Check peripheral address or STEP 5 program

6101	**Illegal MC 5 code**
Cause: Effect: Remedy:	STEP-5 instruction cannot be interpreted PLC STOP Check or reload PLC program Evaluate ISTACK

6102	**Illegal MC 5 parameter**
Cause: Effect: Remedy:	Illegal parameter type (I, Q, F, C, T) or illegal parameter value PLC STOP Check PLC program Evaluate ISTACK

6103	Transfer to missing DB
Cause:	L DW or T DW without prior ”opening” (A DB ...) of a data block
Effect:	PLC STOP
Remedy:	Check PLC program

6104	Substitution error
Cause:	Parameterization error in BMW or BDW command
Effect:	PLC STOP
Remedy:	Correct PLC program

6105	Missing MC 5 block
Cause:	Unavailable block in control called (OB, PB, SB, FB).
Effect:	PLC STOP
Explanation:	OB2 not available, for example
Remedy:	Enter missing block

6106	DB missing
Cause:	Unavailable data block in control called
Effect:	PLC STOP
Remedy:	Enter missing DB

6107	Illegal segment LIR / TIR
Cause:	LIR: Segment No. 0-A allowed TIR: Segment No. 0-6 allowed
Effect:	PLC STOP
Explanation:	Programming Guide LIR/TIR
Remedy:	Correct program

6108	**Illegal segment block transfer TNB / TNW**
Cause: Effect: Explanation: Remedy:	Source : Segment No. 0-A allowed Destination : Segment No. 0-6 allowed PLC STOP See Programming Guide TNB / TNW Correct program

6109	**Overflow - BSTACK**
Cause: Effect: Explanation: Remedy:	Nesting depth of more than 12 PLC STOP When a block calls itself, for example Correct program

6110	**Overflow - ISTACK**
Cause: Effect: Explanation: Remedy:	More than two ISTACK entries PLC STOP Cyclic program (OB1) interrupted by interrupt handler (OB2) and interrupt handler interrupts itself See processing delay OB2, Alarm 6162.

6111	**MC 5 instruction STS**
Cause: Effect: Explanation:	STS instruction programmed in FB PLC STOP Immediate termination of STEP 5 program processing

6112	MC 5 - command STP
Cause: Effect: Explanation:	STP instruction programmed PLC STOP PLC STOP after termination of STEP 5 program processing

6113	Illegal MC 5 timer / counter
Cause: Effect: Remedy:	STEP 5 timer or counter not available or not enabled via MD. Time with a constant of 10 ms progr. PLC STOP. • Correct program, correct time constant (100 ms, 1 s, 10 s) • Change PLC MD 6

6114	Function macro
Cause: Effect: Remedy:	Error in use of a function block (basic program) PLC STOP See Programming Guide, Function Macros Evaluate ISTACK (error number in ACCU 2)

6115	System commands disabled
Cause: Effect: Remedy:	Programmed command LIR, TIR, TNB, TNW PLC STOP Set PLC MD 2003 bit 4

6116	MD 0000: Alarm byte No.
Cause: Effect: Remedy:	PLC MD 0 set greater than 31 PLC STOP Correct MD

6117	**MD 0001: CPU load**
Cause: Effect: Remedy:	PLC MD 1 greater than 20 % PLC STOP Correct MD

6118	**MD 0003: Alarm runtime**
Cause: Effect: Remedy:	PLC MD 3 greater than 2500 µs PLC STOP Correct MD

6119	**MD 0005: Cycle time**
Cause: Effect: Remedy:	PLC MD 5 greater than 320 ms PLC STOP Correct MD

6121	**MD 0006: Last MC 5 time**
Cause: Effect: Remedy:	PLC MD 6 greater than 31 PLC STOP Correct MD

6122	**This arrangement n. permitted**
Cause: Remedy:	A wrong coupling area position (0) was set at the link module master PLC by DIP-FIX (S6). Set DIP-FIX (S6) properly

6123	Illegal servo sampling time
Cause: Effect: Remedy:	NC MD 155 greater than 5 PLC STOP Correct MD

6124	Gap in MC 5 memory
Cause: Effect: Remedy:	Legal and illegal blocks do not follow on smoothly PLC STOP Overall reset and reload PLC program

6125	Inputs assigned twice
Cause: Effect: Remedy: Explanation:	Same address for central and distributed inputs PLC STOP Check addressing for input modules Address central I/O device before distributed device

6126	Outputs assigned twice
Cause: Effect: Remedy: Explanation:	Same address for central and distributed outputs PLC STOP Check addressing for output modules Address central I/O device before distributed device

6127	Alarm byte missing
Cause: Effect: Remedy:	Selected interrupt input byte not available in hardware PLC STOP • Change PLC MD 0 • Adjust address decoding for interrupt byte

6128	Wrong I/O jumpering
Cause: Effect: Remedy:	Same address for central and distributed I/O device PLC STOP Change address decoding

6130	Synch. error basic program
Cause: Effect: Remedy:	Synchronization pattern for assembler function blocks no longer correct PLC STOP PLC OVERALL RESET, reload PLC program if necessary

6131	Synch. error MC 5 program
Cause: Effect: Remedy:	Synchronization pattern for STEP 5 program blocks no longer correct PLC STOP PLC OVERALL RESET, reload PLC program

6132	Synch. error MC 5 data
Cause: Effect: Remedy:	Synchronization pattern for STEP 5 data blocks no longer correct PLC STOP PLC OVERALL RESET, reload PLC program

6133	Illegal block basic program
Effect: Remedy:	PLC STOP Replace system software

6134	Illegal block MC 5 program
Effect: Remedy:	PLC STOP PLC OVERALL RESET, reload PLC program

6135	Illegal block MC 5 data
Effect: Remedy:	PLC STOP PLC OVERALL RESET, reload PLC program

6136	Sumcheck error MC 5 block
Effect: Remedy:	PLC STOP PLC OVERALL RESET, reload PLC program

6137	Sumcheck error basic program
Effect: Remedy:	PLC STOP Replace system software

6138	No response from EU
Cause:	• No operating voltage at EU • Cable to EU not connected or defective
Effect:	POWER ON : PLC STOP, control ramps up automatically after fault has been rectified Cyclic : PLC STOP, as a function of PLC-MD 2003 bit 2
Explanation:	May also occur in the event of major interference
Remedy:	Check cable, address setting EU No. = 0; +24V supply voltage OK?

6139	EU transmission error
Cause: Effect: Remedy:	Incorrect protocol between EU and central controller (NC). See 6138 Check cable; observe installation instructions for fibre-optics cables; shielding (interference)

6140	Impermissible warm restart
Cause: Effect: Remedy:	Problems in operating system after voltage dip PLC STOP Start control with cold restart

6143	Decoding DB not available
Cause: Effect: Remedy:	DB 80 missing PLC STOP Input DB 80

6144	Decoding DB not modulo 6
Cause: Effect: Remedy:	DB 80 must have 3 DWs per extended M function PLC STOP Number of DWs in DB must be a multiple of 3 (3 DWs, 6 DWs, 12 DWs etc.)

6145	Wrong number of decoding units
Cause: Effect: Remedy:	Possible number of decoding units 2, 4, 8, 16, 32 PLC STOP Number of M functions entered in DB must be 2, 4, 8, 16 or 32

6146	**Decoding DB too short**
Cause: Effect: Remedy:	DB 80 not set up over full length (DW 0 - 95) PLC STOP Set up DB 80 in start-up procedure or input subsequently

6147	**Distributed I/Os changed**
Cause: Effect: Explanation:	Module(s) withdrawn or inserted in cyclic operation PLC STOP dependent on PLC MD 2003 bit 2 (Intended for installation)

6148	**Overtemperature in EU**
Cause: Effect: Explanation: Remedy:	Temperature rise in EU, fan breaks down PLC message 6148 displayed Monitor on CC interface module 6FX1132-1BA01 module Check fan

6149	**Stop via softkey PG**
Cause: Effect: Remedy:	Stop command via PG PLC STOP • PLC Start via PG • POWER ON

6150	**Timeout: MC5 user**
Cause: Effect: Remedy:	PLC STOP (S5 prog.) Evaluate fine coding of errors, see Programming Guide

6151	**Timeout: Link memory**
Cause: Effect: Remedy:	PLC STOP Only when master PLC link is used (not currently implemented) Check hardware

6152	**Timeout: LIR / TIR**
Cause: Effect: Explanation: Remedy:	Unavailable addresses accessed PLC STOP See Programming Guide Check segment and offset address. Hardware available?

6153	**Timeout: TNB / TNW**
Cause: Effect: Explanation: Remedy:	Programming error or incorrect use of TNB/TNW PLC STOP See Programming Guide Check reliability of source and destination addresses Addresses available

6154	Timeout: LPB / LPW / TPB / TPW
Cause: Effect: Remedy:	Load/transfer command to faulted I/O device PLC STOP Check I/O device or replace modules

6155	Timeout substitution command
Effect: Remedy:	PLC STOP Check PLC program

6156	Timeout not interpretable
Cause: Effect: Remedy:	No acknowledgement (Timeout) not definable by system program PLC STOP Evaluate fine error diagnostics PLC OVERALL RESET, reload PLC program

6157	Timeout: JU FB /JC FB
Cause: Effect: Remedy:	Unavailable addresses accessed in resident function macros PLC STOP Check hardware

6158	**Timeout with I / O transfer**
Cause: Effect: Explanation: Remedy:	Central I/O device no longer signalling PLC STOP All I/O modules detected on start-up (If the no. of I/O modules changes in cyclic operation this alarm appears) No alarm on failure of +24 V supply! Check bus connections to I/O submodules

6159	**Time exceeded STEP 5**
Cause: Effect: Explanation: Remedy:	Maximum runtime in PLC MD 1 exceeded PLC STOP, dependent on PLC MD 2003 bit 1 Evaluate diagnostic DB • Increase MD 1 CPU load increases • Set MD 2003 bit 6 cycle time increases • Time-optimize PLC program

6160	**Runtime exceeded OB2**
Cause: Effect: Explanation: Remedy:	Maximum runtime in PLC MD 3 exceeded PLC STOP, dependent on PLC MD 2003 bit 0 Evaluate diagnostic DB • Increase MD 3 • Time-optimize OB2

6161	**Cycle time exceeded**
Cause: Effect: Remedy:	Maximum runtime in PLC MD 5 exceeded PLC STOP • Increase MD 5 • Time-optimize PLC program

6162	**Processing time delay OB2**
Cause: Effect: Explanation: Remedy:	The alarm program interrupted itself. PLC STOP, dependent on PLC MD 2003 Bit 0. Evaluate diagnostic DB. Time-optimize OB2, i.e. reduce active processing time of interrupt handler

6163	**Host PLC failure**
	(available soon)

7000 7063	**PLC user messages**
Cause:	Initiation bit was set in the PLC user program

Siemens AG

AUT V250
P.O. Box 31 80
D-91050 Erlangen
Federal Republic of Germany

Suggestions

Corrections

For Publication/Manual:

SINUMERIK 810 GA3
SINUMERIK 820 GA3
Software Version 3
Installation Guide

Installation Instructions

Order No.: 6ZB5 410-0DL02-0AA2
Edition: January 1993

From:

Name

Company/Dept.

Address

Telephone /

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

Siemens AG
Automation Group
Automation Systems
for Machine Tools, Robots
and Special Purpose Machines
P.O Box 31 80, W-8520 Erlangen
(new postcode as from July 1, 1993: D-91050)
Federal Republic of Germany

Siemens Aktiengesellschaft

Order No. 6ZB5 410-0DL02-0AA2
Printed in the Fed. Rep. of Germany
251/332148-14376 IA 06930.2

Progress
in Automation.
Siemens